



DOSRAD

Version 4.5.0

User Manual

February 2012

Document issue: 4



Use of the software and of the present software tutorial is submitted to a license agreement to be accepted before the software installation on a computer.

All suggestion or error concerning the software or this software manual can be sent to:

Airbus Defence and Space

For the attention of Mr M. LEPILLIEZ

Z.I. du Palays

31 rue des Cosmonautes

31402 TOULOUSE CEDEX 4

FRANCE



Contents

Introduction.....	7
About this Document	8
About DOSRAD	9
Glossary	10
Typographic Conventions	11
General Principles.....	12
Overview	13
Introduction	13
DOSRAD-Specific SYSTEMA Tabs	14
Example	14
DOSRAD Modules and Corresponding Files	16
Man-Machine Interface (MMI).....	18
1.1 Introduction.....	19
1.2 In this Chapter.....	21
2 How to Perform a Radiation Analysis	22
3 Using the Interactive MODELER Module.....	23
3.1 DOSRAD Material Tab.....	24
Example	24
3.2 Applicative Properties.....	26
Bulk	26
Related properties	26
Example	27
Activity	27
Configuration	28
Example	28
3.3 Importing a SYSEXP File	29
Example	29
4 Interacting with the Meshing	30
4.1 Dose Cartography.....	30
Meshing.....	30
Numbering.....	32
Example	32



4.2 Bounding Box	33
Meshing.....	33
Example	33
Importing a DOSTAR file	35
Example	35
Example	37
Numbering.....	37
Example	38
Example	38
4.3 Computation Point.....	38
Meshing.....	38
Example	39
Numbering.....	40
Example	40
4.4 Honeycomb specific item.....	41
Meshing.....	41
Numbering.....	41
Dosrad Properties	42
4.5 Volume specific item.....	43
Meshing.....	43
Numbering.....	43
Dosrad Properties	44
5 Using the Interactive MISSION Module	45
6 Using the Interactive PROCESSING Module.....	46
Example	46
6.1 Basic Rule	47
Example	47
Example	49
Example	50
7 Post-Processing	51
Example	51
7.1 How to Display Results in a 3D View	51
Example	51
Computation methods.....	53
8 General Method: Shielding Computation	55
8.1 Overview.....	55
8.2 Honeycomb specific item.....	57
8.3 Volume specific item.....	58
9 Dose Cartography	61



9.1 Theory.....	61
Introduction.....	61
Monte-Carlo ray tracing method.....	61
Surface sampling.....	62
Ray direction sampling.....	63
Computation algorithm.....	63
9.2 Parameters.....	65
9.3 Input / Output.....	66
Input 66	
Output.....	66
10 Bounding Box Analysis.....	67
10.1 Theory.....	67
Introduction.....	67
Bounding box definition.....	67
Computation algorithm.....	67
Discretization of a bounding box mesh.....	69
10.2 Parameters.....	70
10.3 Input / Output.....	71
Input 71	
Output.....	71
11 Computation Point Analysis.....	73
11.1 Theory.....	73
Introduction.....	73
Computation point definition.....	73
Computation algorithm.....	73
11.2 Parameters.....	75
11.3 Input /Output.....	76
Input 76	
Output.....	76
Getting Started with Dosrad.....	77
12 Introduction.....	78
12.1 Overview.....	78
12.2 The DosradSAT Model.....	80
12.3 In this Chapter.....	81
13 DosradSat.....	82
13.1 Introduction.....	82



13.2	Opening an Existing Model	82
	Using a DosradSat model.....	82
13.3	Pre-Processing	84
13.4	Loading Results.....	86
	Loading results related to DosradSat	86
13.5	Post-Processing	86
	Example	87
	Example	88
14	MySat 89	
14.1	Introduction.....	89
	Starting with MySat	89
14.2	Setting the Material References	92
14.3	Completing the Geometrical Model	93
	Example	93
14.4	Creating the Meshing	97
	Initializing the meshing file.....	97
	Dose cartography (or Mapping).....	98
	Bounding box	99
	Computation point	102
	Numbering setting	104
14.5	Creating the Mission	105
	Creating a new mission	105
	Linking the meshing	107
14.6	Defining a Computation Processing Diagram	109
	Creating a processing diagram.....	109
	Mapping module.....	110
	BoundingBox module	110
	ComputPoint module.....	110
	Activating the dose curve file.....	111
	Running the computing files	112
14.7	Results	113
	How to display results in a 3D view	113



Introduction



About this Document

This User Manual is intended for users of the **DOSRAD** application.

It consists of the following chapters:

➤ **Introduction**

Gives a general description of the manual.

➤ **General Principles**

Gives an overview of the **DOSRAD** method and functionalities with the different files and modules.

➤ **Man Machine Interface (MMI)**

Describes all the elements interfaced with the **DOSRAD** application.

➤ **Computation methods**

Presents general computation methods using the different modules:

○ **Dose cartography**

Presents the module describing, at satellite or equipment level, the calculation of the total dose received on model meshes.

○ **Bounding box**

Presents the module describing, at satellite level, the calculation of six Aluminium equivalent thickness of any equipment for subcontractor specification.

○ **Computation point**

Presents the module describing, at satellite or equipment level, the calculation of the total dose at one point in the geometrical model.

➤ **Getting Started with DOSRAD**

Provides step-by-step instructions for getting started with the **DOSRAD** application.



About DOSRAD

DOSRAD is a plug-in application of the **SYSTEMA** environment. This manual makes therefore reference to the modeler, mission and processing modules of **SYSTEMA**.


This document presents the **DOSRAD** software that computes the radiation dose received by electronic components located within the spacecraft accounting for the geometrical shielding and the mission dose depth curve.

This manual contains task-oriented instructions which show you how to use the **DOSRAD** program. The different computation techniques supported by **DOSRAD** are also presented.



Glossary


In this User Manual, some terms have a specific meaning:

Term	Meaning
BULK	This container is used to specify the applicative properties of the surfaces.
Local / inherited	 The "local" and "inherited" notions are explained in the <i>SYSTEMA User Guide</i> (Chapter "Geometrical model management")
Activity	In terms of radiation application, the question is to determine whether a surface has an impact on the dose calculation. An inactive surface does not give shielding to radiation.
Specific Items	The specific items (or virtual nodes) are elements defined in the meshing model, on which results are computed (Bounding box and Computation point) or allowing to extend standard geometrical definition (honeycomb and volumes).
Referenced shapes	When creating a virtual node, the shapes selected for its definition are called referenced shapes. These attached shapes modify the shielding to radiation in the analysis. The referenced shapes by Bounding Box or Computation Point give no shielding to radiation. The referenced shapes by Volumes or Honeycomb modify the way the ray-tracing algorithm will account for thicknesses.
Bounding box	A bounding box is the virtual target box (6 faces box) on which the sectoring analysis will be performed.
Computation point	A computation point is the virtual point on which the Computation point analysis will be performed. It is also the attached point at the bounding box creation.
Honeycomb	A honeycomb is a specific item allowing to take into account additional thickness during ray-tracing. These additional thicknesses come from honeycomb panels present in a spacecraft. A Honeycomb specific item is a pair of referenced shapes representing the two face sheets of a honeycomb panel.
Volume	A volume is a specific item allowing to represent volumes in a geometrical model. A Volume specific item is composed of referenced shapes defining the border of the volume to be represented.



Typographic Conventions

The following typographic conventions are used in this manual:

Convention	Description	Example
Green	Used for the DOSRAD -related functions	The DOSRAD application is interfaced with...
Blue	Used for the SYSTEMA -related functions	Within the SYSTEMA <i>processing</i> tab,....
	These colors are used in the diagram of Chapter Man-Machine Interface to distinguish the SYSTEMA entities from the DOSRAD ones.	
<i>Italics</i>	Names of the options related to the current entities or filenames	Fit – Material Filename* – Physical properties
Numbered list	Procedure containing the several actions to be performed	<ol style="list-style-type: none"> 1. Click the Revert button. 2. Enter a new value.
Bulleted list	Items list	This container will affect the following: <ul style="list-style-type: none"> ➤ Dose cartography ➤ Bounding box analysis ➤ Computation point analysis



A warning calls attention to information that is essential to run successfully the DOSRAD application.



A caution calls attention to important information which, if not understood, will lead to a conflict or an error.



A note provides additional information to emphasize or explain important points in the manual.



Cross-reference: prompts the user to refer to a particular section in the manual or to a special document.

* *Material Filename*" is the name of the material which has been loaded during the **SYSTEMA** session.



General Principles



Overview

Introduction

In this chapter the user will find some general principles that he has to know before using the **DOSRAD** files and modules.

The **DOSRAD** software evaluates the radiation dose expected in a spacecraft given the mission dose-depth curve (computed from in-orbit particle fluxes) and the spacecraft geometry including the shielding design.

Radiation doses result from an incident flux of particles in space, mainly protons and electrons, which then propagate through the spacecraft material, generating secondary radiation, mainly bremsstrahlung photons (X rays) and depositing dose. Such dose calculations are necessary for evaluating the lifetime of electronic components, payload damage and hazards to astronauts. Future space programmes will also be more demanding in terms of resistance to the effects of the space environment. Long-term missions and the use of high-performance technologies all require reliable environment models and analysis tools.

The software is intended to provide a tool for evaluating doses received at points within spacecraft geometry. The physical processes involved in radiation transport are complex and a rigorous analysis would involve Monte-Carlo methods. However for many engineering applications, a more approximate treatment is more efficient.

The main functions of **DOSRAD** are as follows:

- Geometry modeling and physical properties & meshing.
- Physical simulation: radiation analysis:
 - Dose cartography: calculation of the total dose received on model meshes.
 - Bounding box analysis: calculation of the six aluminium equivalent thickness of any equipment for subcontractor specification.
 - Computation point analysis: calculation of the total dose at one point in the geometrical model.



DOSRAD-Specific SYSTEMA Tabs

Here the **DOSRAD** package is fed back to the **SYSTEMA** environment.

The **MODELER**, **MISSION** and **PROCESSING** tabs have the same layout.



A detailed table will be proposed in the relevant MMI related sections to describe each tab.

Example

Here are the options proposed in the **Edit** menu of the **SYSTEMA Mission** tab:



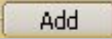
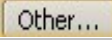

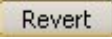
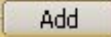
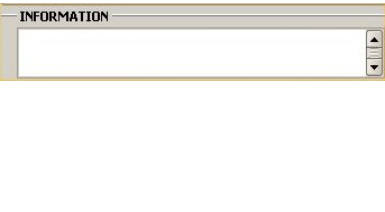
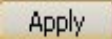
Figure 1 - Options proposed in the **SYSTEMA Mission** tab

For each tab, the menu consists of both general and specific options.



The standard options are as follows:

Edit menu (contains options applicable to entity components)

Option	Used to:
	Enter a specific meshing created for DOSRAD
	Select the file to open
	(Not used in the version 4.3)
	Cancel the display and return to the  option
	Issue a warning or provide information on the processed nodes or the current configuration. The local errors (e.g. a forbidden value) are identified in red. However, some elements cannot be automatically returned in error. If it is not an error to be locally corrected, a warning displays in the lower window. The error is detailed and can be corrected in another area.
	Validate the selected component(s)



The  button is disabled if any error is still present in the information area.



DOSRAD Modules and Corresponding Files

In this User Manual, the "module" or "DOSRAD module" terms will be used to refer to the sub-processes managed by the DOSRAD application, namely *dose cartography module*, *bounding box module*, *computation point module*.

Each module defines its input files, parameters and output files.

The parameters of each module will be detailed in the chapters dealing with the different modules.

The following tables present the different modules and corresponding files:

INPUT File extension

Module	File extension
<i>Computation point</i>	".sysset" ".dose"
<i>Bounding box</i>	".sysset" ".dose"
<i>Dose cartography</i>	".sysset" ".dose"



OUTPUT File extension

Module	File extension
<i>Computation point</i>	“.pt.h5”, “.rpt.h5” “.pt.log” “.pt.out”
<i>Bounding box</i>	“.box.h5”, “.rbox.h5” “.box.log” “.box.out”
<i>Dose cartography</i>	“.map.h5”, “.rmap.h5” “.map.log”



INPUT file extension

The **“.sysset”** file refers to a mission defined in the **SYSTEMA V4** platform.

The **“.dose”** file contains the dose depth curve with different types of geometry (solid sphere, shell sphere, semi infinite wall and finite thickness slab).

OUTPUT file extension

The **“.h5”** outputs are HDF5 files used for result interpretation. They can also be loaded into an HDF explorer.

The **“.log”** file provides general information.

The **“.out”** files output by the bounding box and computation point modules are results written with the ASCII format.



Page left intentionally blank

Man-Machine Interface (MMI)

Overview

1.1 Introduction

This chapter describes the general ergonomic principles and basic procedures for using the **DOSRAD** application into the **SYSTEMA** platform.

The application may be illustrated as follows:

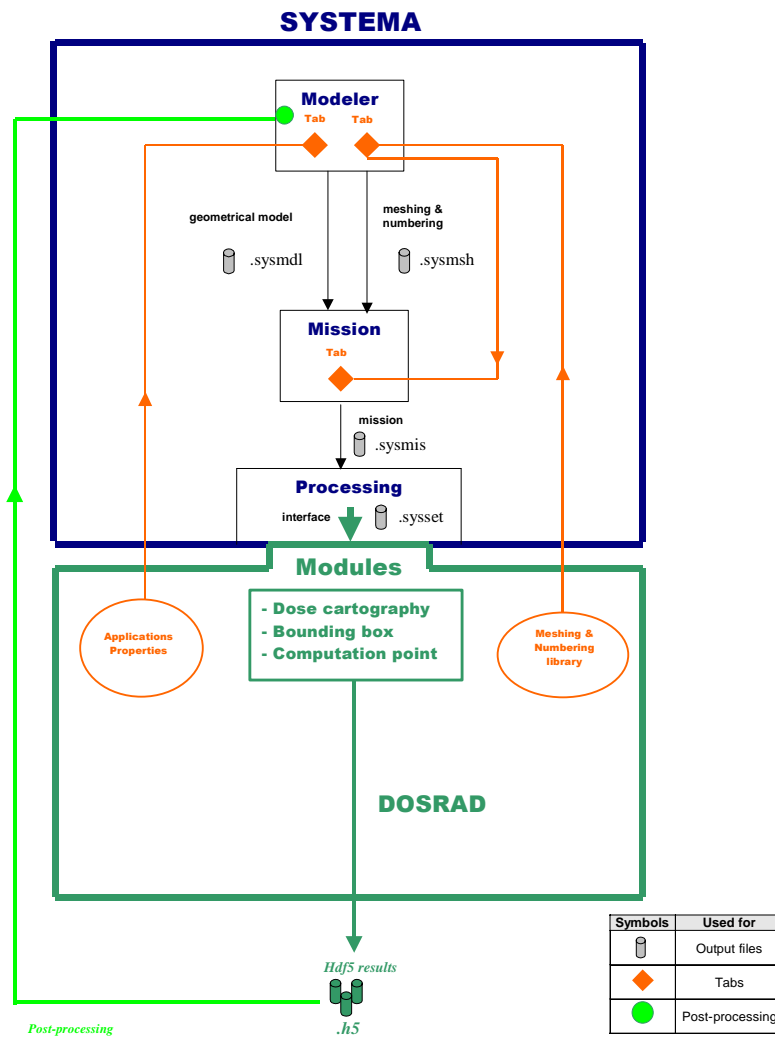


Figure 2 - **DOSRAD** within **SYSTEMA**



Note that, since version 4.4.2, the Mission is not anymore mandatory for setting a calculation as the processing can be directly linked to the meshing file.



1.2 In this Chapter

This chapter covers the following topics:

- How to perform a radiation analysis.
- Using the interactive Modeler module.
- Meshing and numbering.
- Using the interactive Mission module.
- Using the interactive Processing module.
- Post-processing.



2 How to Perform a Radiation Analysis

The **DOSRAD** application is interfaced with **SYSTEMA**. Consequently, to use the **DOSRAD** package, the analysis must start with the generation of the geometrical model of the space system.

This first step of the analysis can be performed:

- Using the interactive **3D MODELER**.

A mission can also be designed (not mandatory anymore since version 4.4.2). This second step can be performed:

- Using the interactive **MISSION** module.

The different steps to build a radiation analysis are performed:

- Using the interactive **PROCESSING** module.



As shown in the illustration (Figure 1 - **DOSRAD** within **SYSTEMA**), each **SYSTEMA** module (Modeler, Mission & Processing) are completed by **DOSRAD** specific tabs.

Within the **SYSTEMA** processing tab, the **DOSRAD** application is organized into modules, which are based on 3 sub-applications and used to manage the **DOSRAD** entities:

- Computation point:** this module is used to calculate the total dose at one point in the geometrical model.
- Bounding box:** this module is used to calculate the six Aluminium equivalent thickness of any equipment for subcontractor specification.
- Dose cartography:** this module is used to calculate the total dose received on model meshes. It gives dose mapping on surface whatever their shape (cone, cylinder, sphere...).



The above-listed modules are shown on (Figure 11 - **DOSRAD** modules).



3 Using the Interactive MODELER Module

The Modeler tab is used to work on models and meshing. According to the **SYSTEMA** configuration, some applications can be interfaced with **SYSTEMA**. In this case, the elements of the geometrical model have application-dependent properties.

These properties can be consulted or modified using the edition window, via a specific tab (called "**DOSRAD**" tab" in this document). Each application has its own tab.

For a given application, material files may be required to define the application-dependent properties. These files are declared at the root level of the geometrical model.



Refer to SYSTEMA User Guide (Chapter 4: "Geometrical Model Management").

3.1 DOSRAD Material Tab

The user defines material associations via the specific tab of the edition window. **DOSRAD** is provided with a default reference material database. However, the user may want to use its own material database. In the following figure, the user can associate a material file with the application-dependent properties.

Example

The Material tab (Figure 2 - **DOSRAD** material tab) lets the user enter a material filename for the physical properties.

To get access to this window, open the edit window at the root level and unselect "Use default material database".

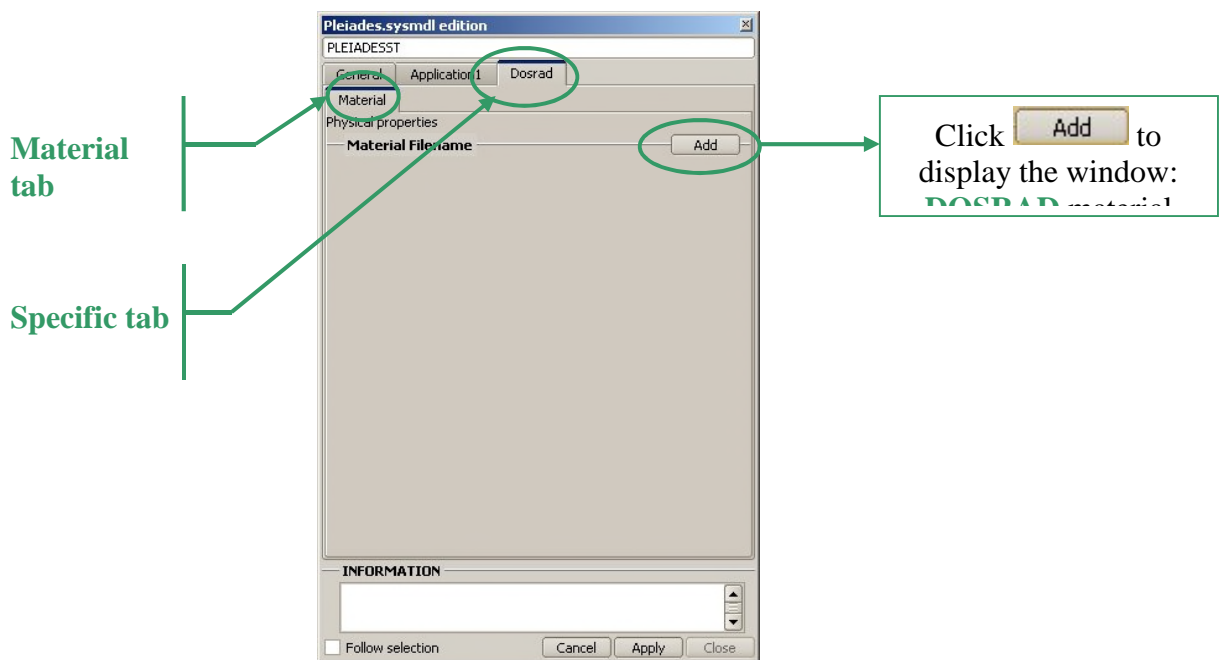
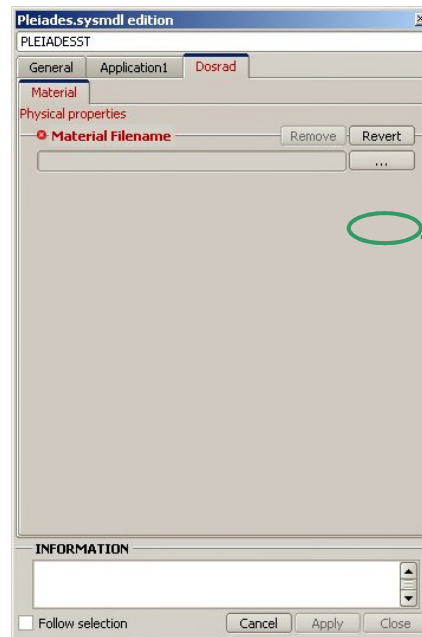


Figure 3 - **DOSRAD** material tab




Click  to select a file

Figure 4 - Material filename



The “.sysmtr” file contains densities of the materials used in the model (given in kg/m3). A material editor accessible from **material sub-tab** in the **SYSTEMA** modeller tab allows to create or modify the material file. A density is associated with each material defined in this file. The materials are visible as a green bulleted list.



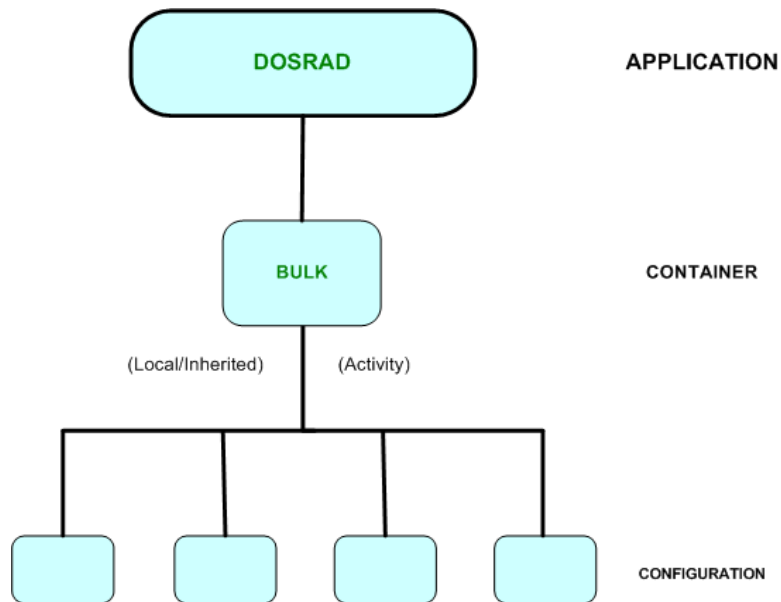
DOSRAD is delivered with a reference material database. By default, all newly created models will be linked to this reference material database. Use the above procedure to attach or create a user defined material database.



Since version 4.4.2, the material file format has changed. Before version 4.4.2 the file extension was “.sysmat”. Opening an existing “.sysmat” file is still possible; it will be automatically converted to the new format.



3.2 Applicative Properties



Applicative properties are specified in the BULK container. This container (BULK) includes property sub-groups.

Some of these sub-groups may globally be assigned by a material.

Bulk

This container refers to the internal material settings.

The following modules are impacted:

- Dose cartography.
- Bounding box.
- Computation point.

Related properties

The properties linked to this container are as follows:

- **Geometrical properties (Thickness).**
- **Physical properties (Density).**



Example



Figure 5 - Object edition

Activity

Here, the question is:

- **Is the surface taken into account in the radiation calculation?**

If the user wants the surface to be taken into account in the radiation calculation, the Activity has to be set to “yes”. Otherwise, the Activity has to be set to “no”. The activity is thus the answer to the question.



Configuration



Refer to *SYSTEMA User Guide (Chapter 4: "Geometrical Model Management")*.

The tabs in **DOSRAD** have inheritance properties which are identical to those of **SYSTEMA**. It is thus possible to redefine a configuration locally (inheritance removed).

Example

You can enter the value at the higher level and make the next level inherit this value (default behavior).



The parameter values can be modified even when in an inheritance configuration.

In order of priority, a parameter may be affected by:

- Value overloading (local value).
- Material overloading.
- Inherited value (value inherited from the father).
- Default value.



The customization option 'Overload' lets the user overload the material's default value.



Refer to *SYSTEMA User Guide (Chapter 4: "Geometrical Model Management")*.

3.3 Importing a SYSEXP File

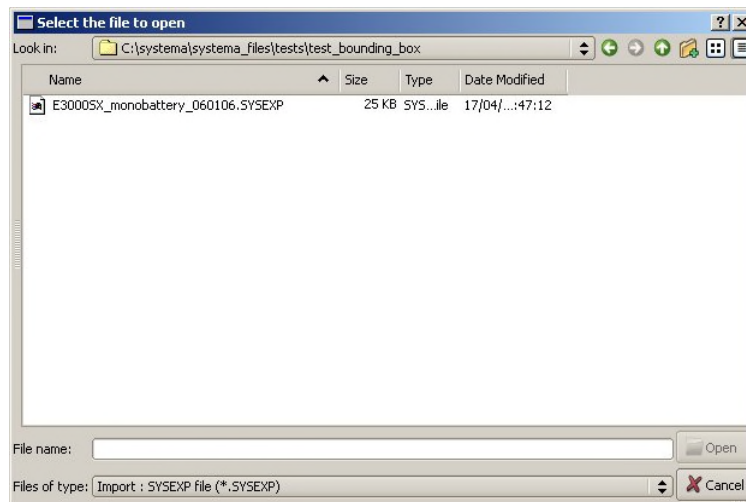
This function allows files (file.SYSEXP) from a previous **DOSRAD** application (version 3) to be imported and to be compatible with a new **DOSRAD** application (since version 4.3.0).

To open the existing model:

1. Open the **Modeler** tab in **SYSTEMA**.
2. To open the existing model, click the **File** -> **Open** menu.

The **Select the file to open** dialog box opens:

Example



3. In "Files of type", select **Import: SYSEXP file (*.SYSEXP)**.
4. Open the "*.SYSEXP" file.

The model is displayed in the 3D view in the **DOSRAD** application.

You can work with the file.

5. Click the **File** -> **Save** menu to save the file as **.sysmdl** (V4 format).



4 Interacting with the Meshing

The way of meshing the model depends on the analysis that will be performed.

For a dose cartography, a region of the model (or the entire model) will be meshed at surfaces level.

For a bounding box computation, the meshing consists on creating one or several bounding boxes (specific items) which are then meshed.

For a computation point analysis, the meshing consists on creating one or several computation points (specific items).

The meshing may also contains additional information allowing to modify the way the ray-tracing algorithm computes thicknesses when encountering modeled shapes. These additional information are stored in specific items, either Volume specific items or Honeycomb specific items. In this case, the user has to provide **DOSRAD** properties to the created specific items.

4.1 Dose Cartography

Meshing

The meshing is associated with an inheritance property and can be accessed from the tree structure.

- Application name:
Determines the application for which the meshing is performed (useful in case **DOSRAD is not the sole installed application**).
- Meshing type:
An application may have one or several different processes of meshing surfaces.



Refer to *SYSTEMA User Guide (Chapter 4: "Geometrical Model Management")*.



The three parameters: "a", "b", "c" define the number of desired intervals in a specific direction.

- Parameters "a" & "b": refer to all two-dimensional elemental surfaces.
- Parameter "c": refers to three-dimensional surfaces.

The following shapes may be distinguished:

- Triangle, rectangle, quadrangle

Triangle	Rectangle	Quadrangle

- Disk, cylinder, cone, parabola, antenna, sphere

Parameter	Segmentation based on:
a	angular sector

Parameter	Shape	Segmentation based on:
b	disk	radius
	cone, cylinder	height
	sphere, parabola	curvilinear abscissa in iso
	antenna	radius

- Polygons
 - The meshing corresponds to the number of desired intervals in a specific direction. This does not apply to the polygon which is a particular case because it is divided in triangles and, consequently, only the “a” parameter will have an impact as regards the detail of surface triangulation.
- Shape macros and shapes with third dimension
 - Parameters “a” & “b” refer to the basic meshing (see above described categories)
 - The “c” parameter completes the meshing in the third dimension.

Numbering

It consists in numbering the surfaces for assigning a number to meshes and defining an increment value. Actually, it is an incrementation used for the global numbering of the model or a sub-part of the model to obtain a mesh network with a meaningful numbering system.



SYSTEMA provides an automatic numbering sufficient for a **DOSRAD** calculation.



Inheritance notion: for a numbering, an incrementation is defined.

Example

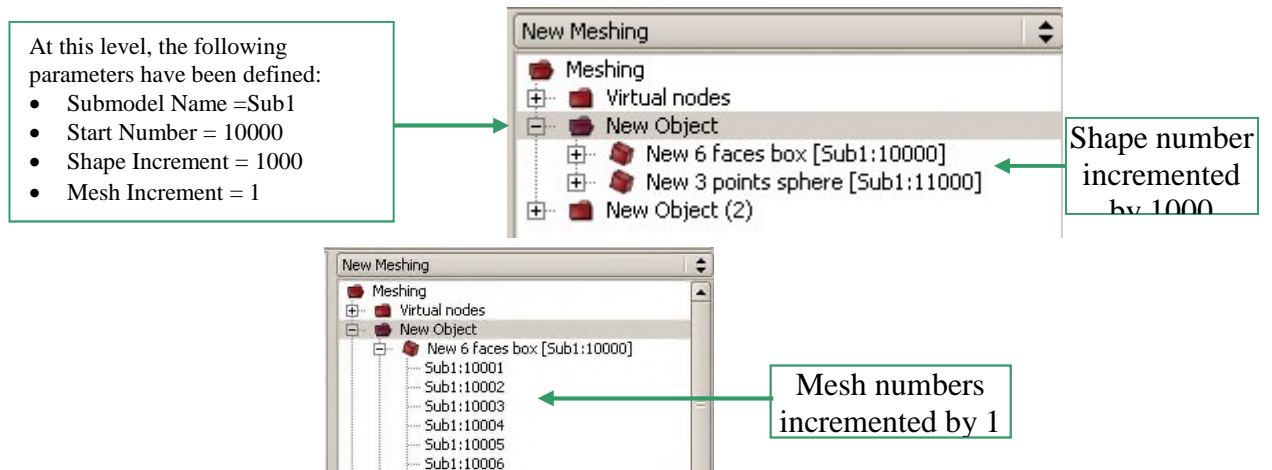


Figure 6 - Inheritance principle for numbering example



4.2 Bounding Box

Meshing

The bounding box is a virtual box. This virtual box is the target on which sectorial analysis will be run (bounding box module).

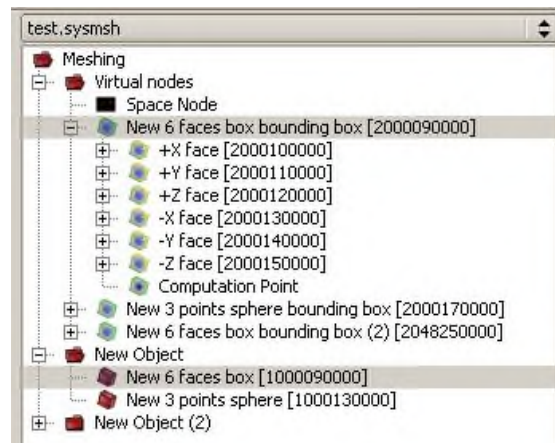
The bounding box is defined in the **meshing model** from one or several shapes of the geometrical model.

To create a bounding box from selection:

1. Open the required meshing model.
2. Select one or several shapes of the geometrical model in which you wish to create a bounding box.
3. Via the right click menu, select **Create bounding box from selection**.

Result: The bounding box is created. In the browser window, the bounding box is positioned under **Specific Items**. Its name finishes with "bounding box". One computation point is systematically added to this bounding box, located at the center and with the same referenced shapes.

Example



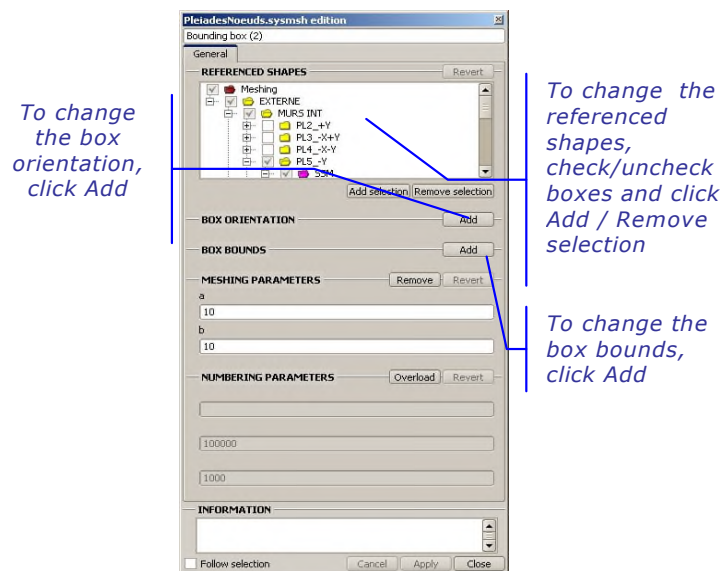
All parameters describing the virtual target box are defined via a specific dialog box.

The following properties, associated with the bounding boxes, could be modified by the user:

- Referenced shapes.
- Computation point.
- Box orientation (by default, the axes are parallel to the system frame).
- Box bounds (the created box is the smallest box including the referenced shapes).
- Meshing parameters (the default values are 5*5*5).

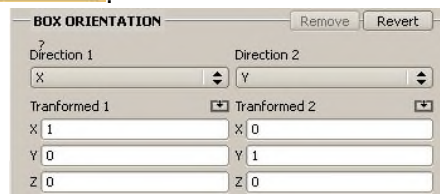
How to change the bounding box orientation, size, meshing and referenced shapes

1. In the browser window, select the required bounding box.
2. In the right click menu, select **Edit**.



To change the box orientation

1. Click **Add**.



2. Change the box orientation parameters.




To change the box bounds

1. Click .

2. Change the box bounds parameters.

To validate the modifications

- Click .

Importing a DOSTAR file



Only available for the bounding boxes.

This function allows files (file.DOSTAR) from a previous **DOSRAD** application (version 3) to be imported and to be compatible with a new **DOSRAD** application (since version 4.3.0).

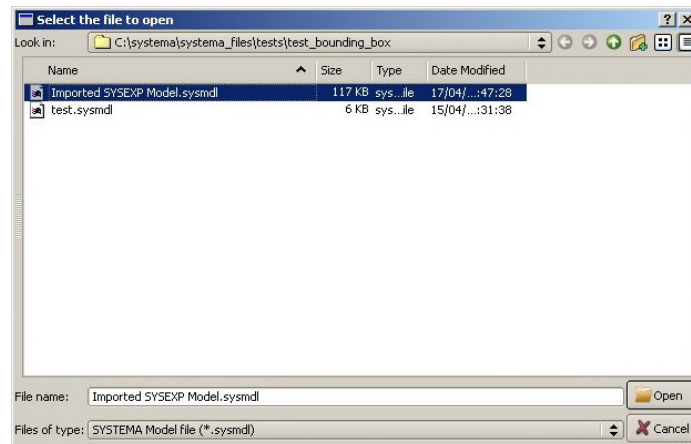
When importing a DOSTAR file, the new bounding boxes are created with the same dimensions, positions and meshing. The analysis point is set at the center of the box by default, but it can be modified. The names of the new bounding boxes are not imported.

The creation of a new meshing based on a DOSTAR file is described as follows:

1. Open the **Modeler** tab in **SYSTEMA**.
2. To open the existing model, click the **File -> Open** menu.

The **Select the file to open** dialog box opens:

Example



3. Select the “*.sysmdl” file.

4. Click **Open**.

The model is displayed in the 3D view in the **DOSRAD** application.

5. Open a new meshing by clicking the **File -> Meshing-> New** menu.

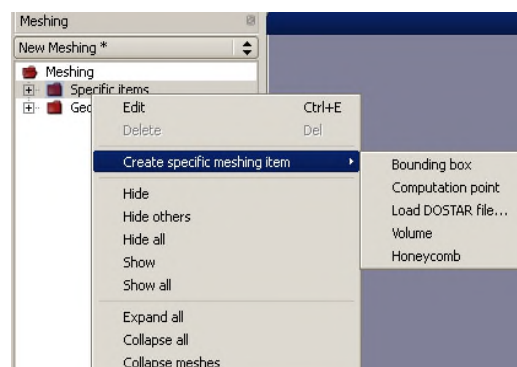
6. In the Meshing browser window, double click **Meshing** to edit and set the meshing parameters:

- Application name (**DOSRAD**).
- Meshing parameters.

7. Click **Apply**.

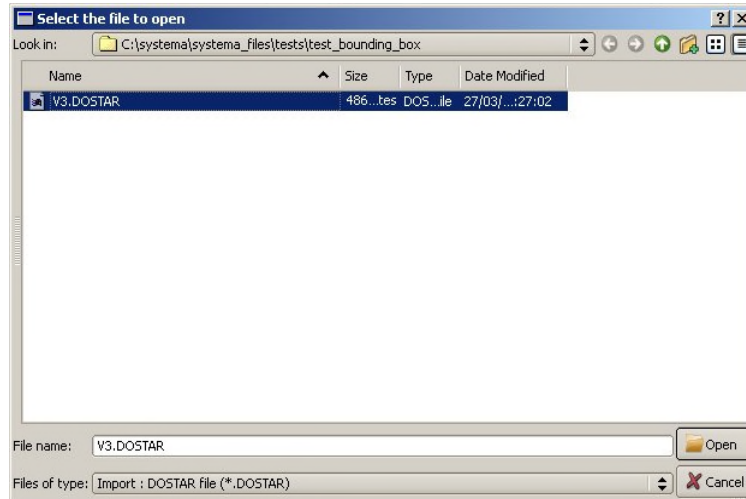
8. In the Meshing browser window, select **Specific Items**.

9. To load the DOSTAR file, via the right click menu, select **Load DOSTAR file....**



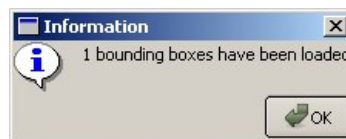
The **Select the file to open** dialog box opens:

Example



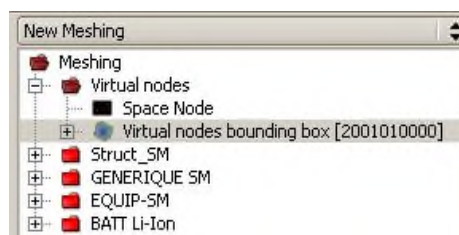
10. Click **Open**.

The following information is displayed:



11. Click **OK**.

The DOSTAR file has been imported in the **DOSRAD** application (version 4).



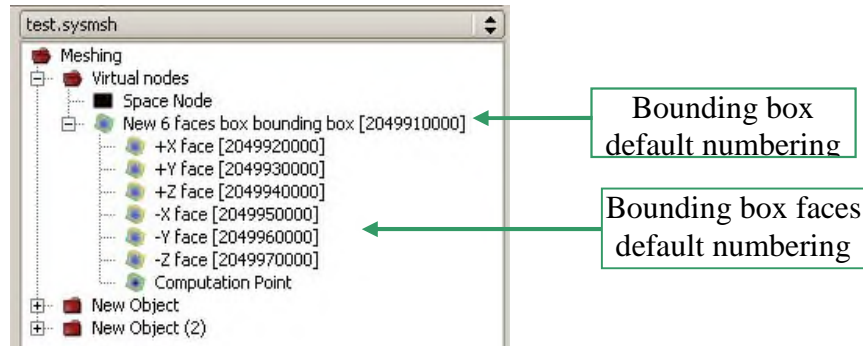
12. Click the **File -> Save** menu to save the file as **.sysmsh** (V4 format).

Numbering

At the bounding box creation, a default numbering is performed by **SYSTEMA**.

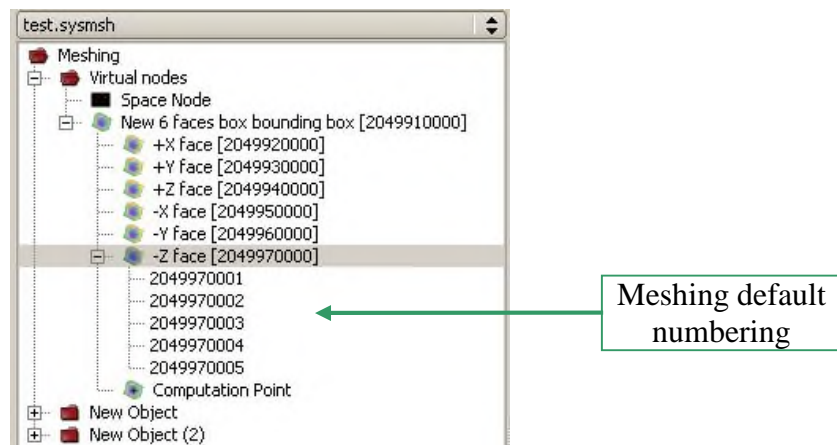
- Default numbering allocated to the bounding box.
- Default numbering allocated to the bounding box faces.

Example



When the bounding box meshing is performed, a default numbering is allocated to the meshing by **SYSTEMA**.

Example



4.3 Computation Point

Meshing

The virtual point is the point on which analysis will be run (computation point module).

The computation point is a virtual point defined in the **meshing model** by its position and referenced shapes.

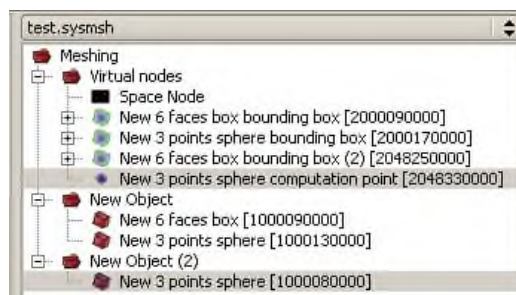
To create a computation point from selection:



1. Open the required meshing model.
2. Select one or several shapes of the geometrical model in which you wish to create a computation point.
3. Via the right click menu, select **Create computation point from selection**.

Result: The computation point is created. In the browser window, the bounding box is positioned under **Virtual nodes**. Its name finishes with "computation point".

Example

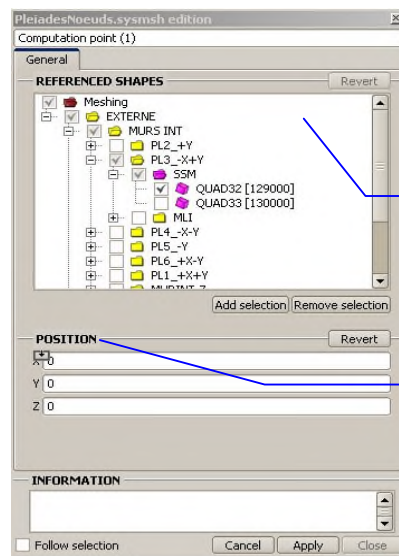


The following properties, associated with the computation point, could be modified by the user:

- Referenced shapes.
- Computation point position (the default value is 0,0,0).

How to change the computation point position and referenced shapes

1. In the browser window, select the required computation point.
2. In the right click menu, select Edit.



To change the referenced shapes, check/uncheck boxes and click Add / Remove selection

To change the computation point position

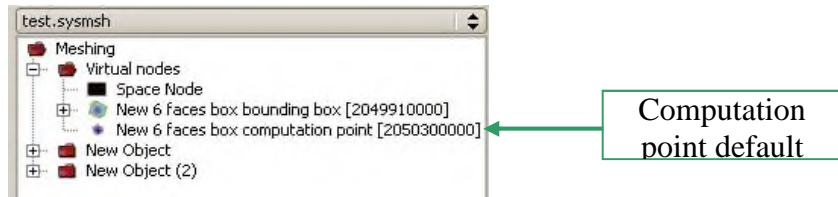


3. To validate the modifications, click  .

Numbering

At the computation point creation, a default numbering is performed by **SYSTEMA**.

Example



4.4 Honeycomb specific item

Meshing

The honeycomb specific item is a way to account for honeycomb core panels.

It is defined in the **meshing model** by two set of referenced shapes which must be the two face sheets of the panel.

To create a honeycomb from selection:

1. Open the required meshing model.
2. Select one or several shapes of the geometrical model representing one of the two panel face sheet.
3. Via the right click menu, select **Create specific meshing item -> honeycomb**.
4. A window opens (see Figure below). In the General Tab, in the Face 2 part, select the shapes of the geometrical model representing the second panel face sheet

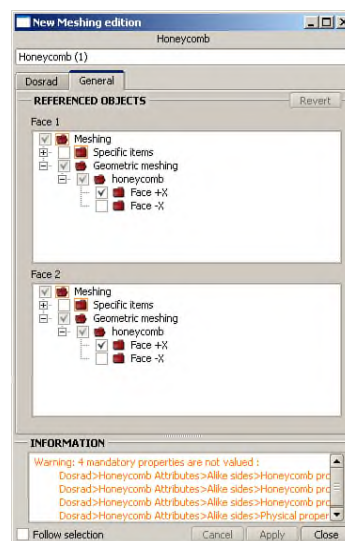


Figure 7 - Honeycomb referenced shapes setting

5. Set the honeycomb properties in the Dosrad tab: nominal panel thickness, foil thickness, cell size and density.

Numbering

At the honeycomb creation, a default numbering is performed by **SYSTEMA**.



Dosrad Properties

The properties of the Honeycomb specific item are the following:

Property	Unit
Nominal Panel Thickness	m
Foil thickness	1/10000 inches
Cell size	inches
density (of the foils)	kg/m ³



The nominal panel thickness includes the two face sheets thicknesses. It is the sum of the honeycomb core thickness and the two sheets thickness.



For a geometrical description of the properties, see Figure 2 in the “Computation Methods” section of this manual.

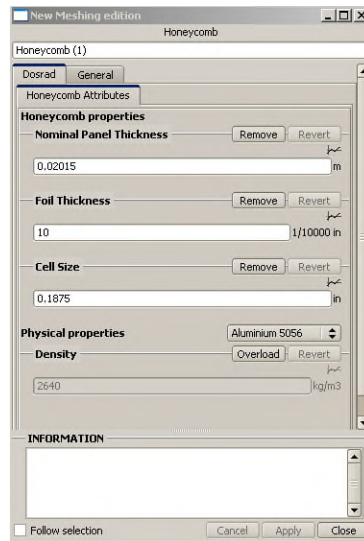


Figure 8 - Honeycomb properties setting

4.5 Volume specific item

Meshing

The volume specific item is a way to account for geometrical entities as plain volume (with difference to standard shell representation).

It is defined in the **meshing model** by a set of referenced shapes which must be the border of the volume to represent.

To create a volume from selection:

1. Open the required meshing model.
2. Select one or several shapes of the geometrical model representing the border of the volume to account for.
3. Via the right click menu, select **Create specific meshing item -> volume**.
4. A window opens (see Figure below). In the General Tab, it can be checked that the correct shapes are referenced.



Figure 9 - Volume referenced shapes setting

5. Set the volume property in the Dosrad tab: the density of the volume has to be input.



The fact that the shapes referenced by a volume are effectively representing the border of a volume is not automatically checked. The user shall thus verify this by visual inspection (no hole).

Numbering

At the volume creation, a default numbering is performed by **SYSTEMA**.

Dosrad Properties

The Dosrad property of a Volume specific item is simply the density of the volume to be represented. For instance, if the Volume is a box representing a complex equipment, the density shall be such that the right mass is accounted for.

The density can be set either by selecting one of the material defined in the material file or by inputting directly a density.



Figure 10 - Volume density setting



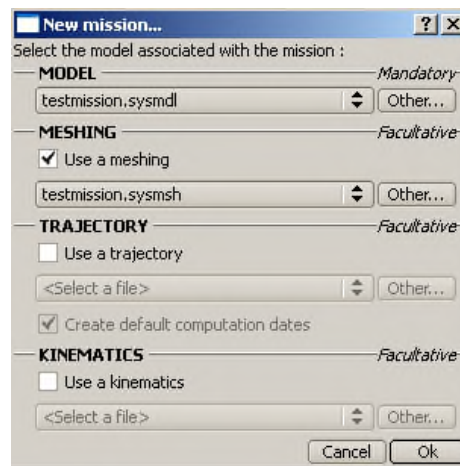
5 Using the Interactive MISSION Module

This Mission module is a basic association module (Geometrical model, Meshing model) making up the application inputs. Since version 4.4.2 it is not mandatory to build a mission to perform a radiation analysis.

It is still possible however to build a mission and use it to perform the analysis, although there is no added value to do so, because no settings available in the mission will be accounted for by the **DOSRAD** modules.

How to create a mission from a meshing/model pair:

1. Go to the Mission tab
2. File -> New, the following window opens



3. Either click OK if the suggested model and mission are the one defining the desired computation or select the relevant model and/or meshing and then click OK
4. The mission is set and will be available in the processing tab to be attached to a process.



6 Using the Interactive PROCESSING Module

This module is composed of a sequence of processes, which are specific to the associated application.

A calculation is defined (mission or meshing selected) and performed (run).

The analysis that will be performed using the mission or meshing will output the results based on **Mapping**, **BoundingBox** and **ComputePoint** modules.



Refer to *SYSTEMA 4.4 User Guide (Chapter 8: "Process Management")*.



The modules are not detailed at this stage. In this User Manual, you will find chapters dedicated to the different modules.

Example

1. Open the **Processing** tab in **SYSTEMA**.
2. To open a new processing, click **File -> New** menu.

Here are the menus proposed in the **SYSTEMA Processing** tab:

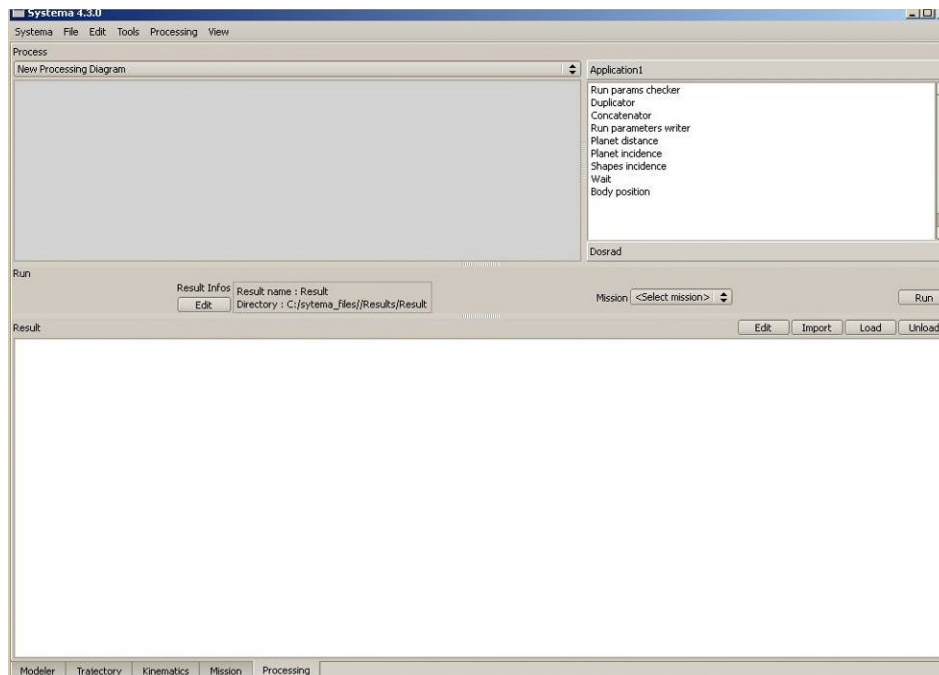


Figure 11 - **SYSTEMA** Processing tab display

6.1 Basic Rule

To display the **DOSRAD** available processes on the process browser:

- Click **DOSRAD**.

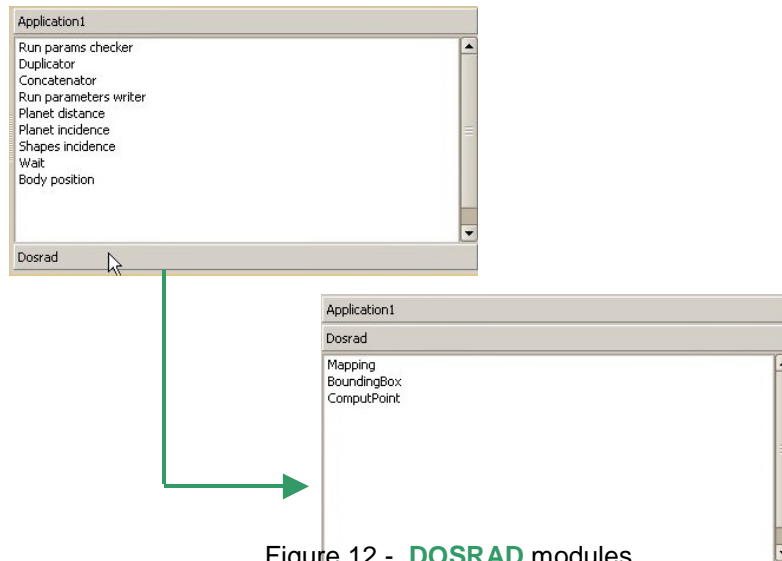


Figure 12 - **DOSRAD** modules

To display a **DOSRAD** module:

- Double click (or Drag and Drop) the selected **DOSRAD** module (Figure 12 - **BoundingBox** module).

Example

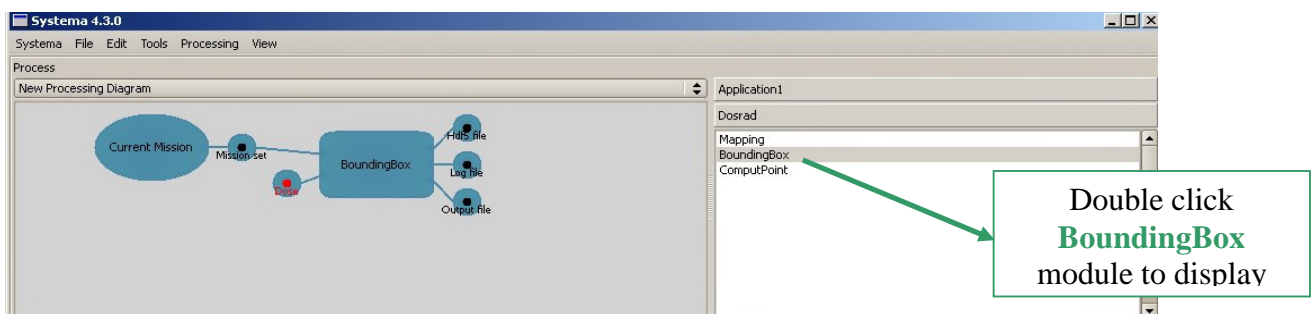


Figure 13 - **BoundingBox** module

To display all the **DOSRAD** modules, the user has to repeat the task shown on (Figure 12 - **BoundingBox** module). The following window displays:

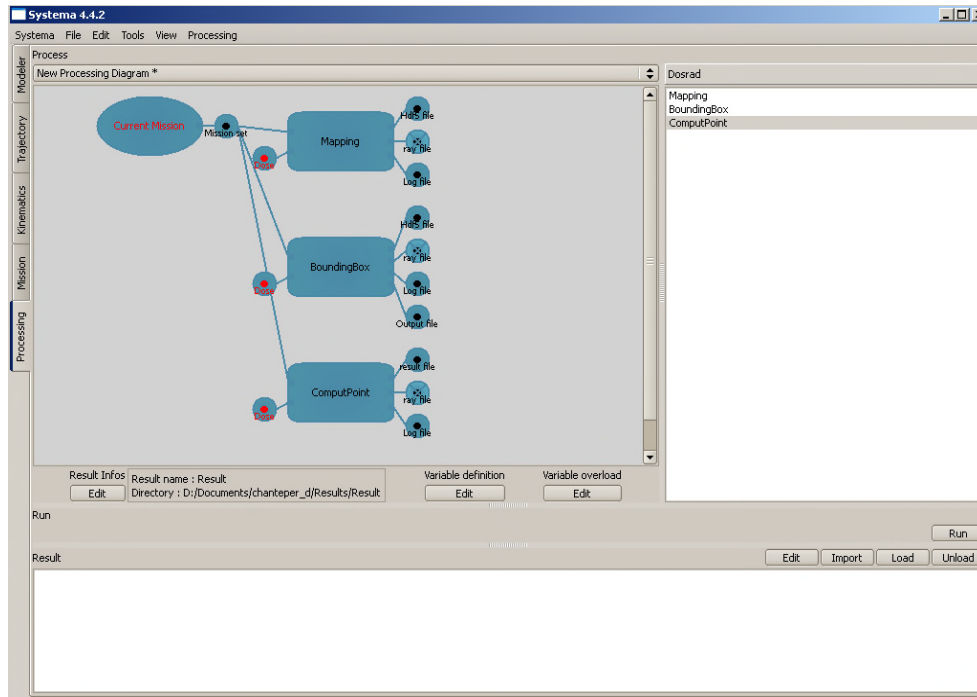
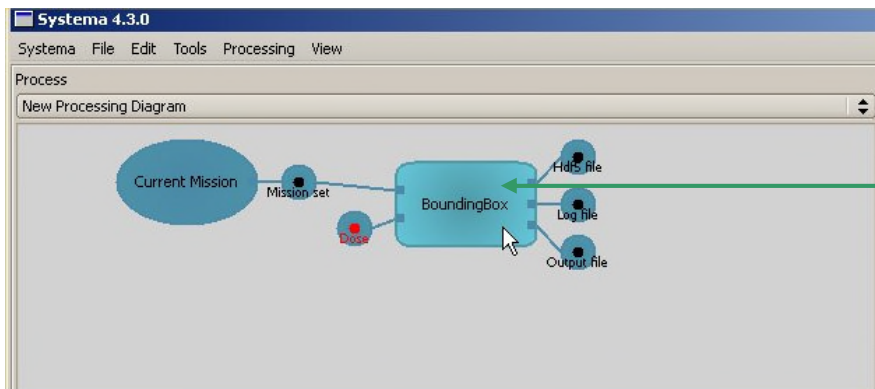


Figure 14 - **DOSRAD** modules

To display the edit or delete option for each **DOSRAD** module, as shown on the following window, the user can perform the following actions on the selected **DOSRAD** module:

- Double click.
- Right click.



A double click or a

Figure 15 - How to get access to the Edit or Delete option of a **DOSRAD** module

Example

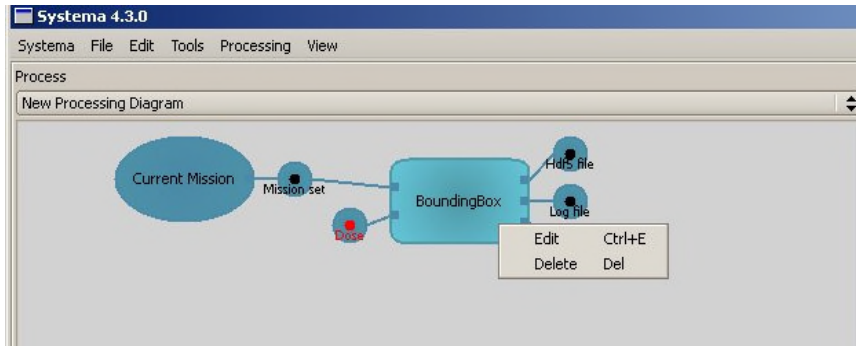


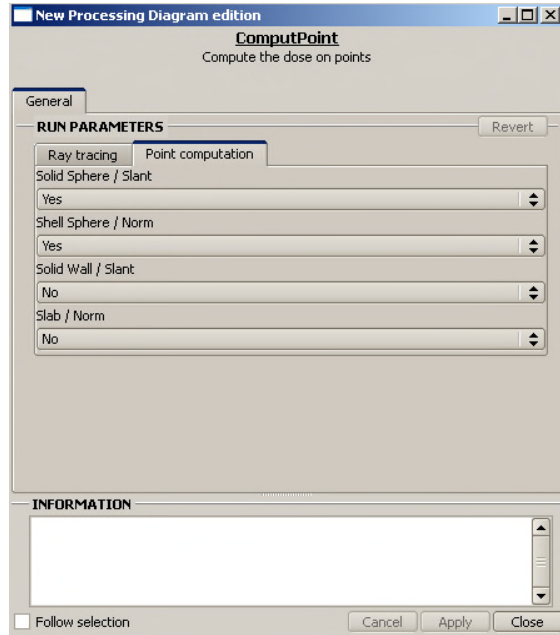
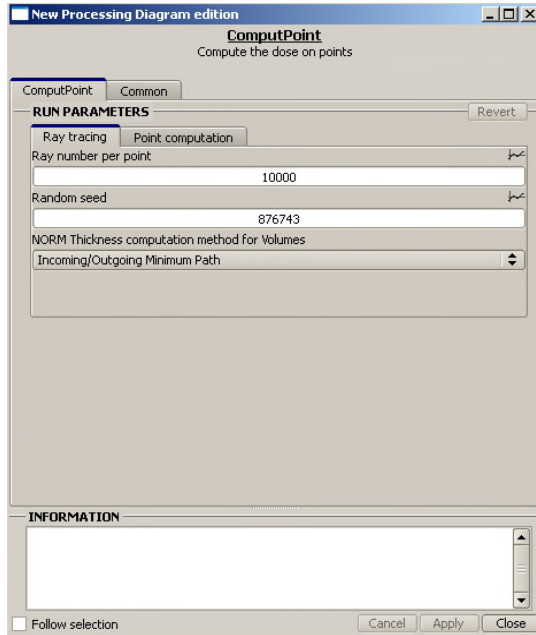
Figure 16 - Edit or delete option of the **DOSRAD BoundingBox** module

- Using the delete option on any **DOSRAD** module enables the user to make this module disappear.
- As regards the edit option, each **DOSRAD** module selected has its own run parameters.
The run parameters have always at least two tabs:
 - **Ray tracing with its own options depending on the selected module. All three modules share at least the choice of the NORM thickness for volume method.**
 - **“Module” computation: “Module” refers to BoundingBox, Mapping or ComputPoint depending on the selected module. The “Module” computation options are the same for the three modules.**



Example

Run parameters available for the **ComputPoint** Module



7 Post-Processing

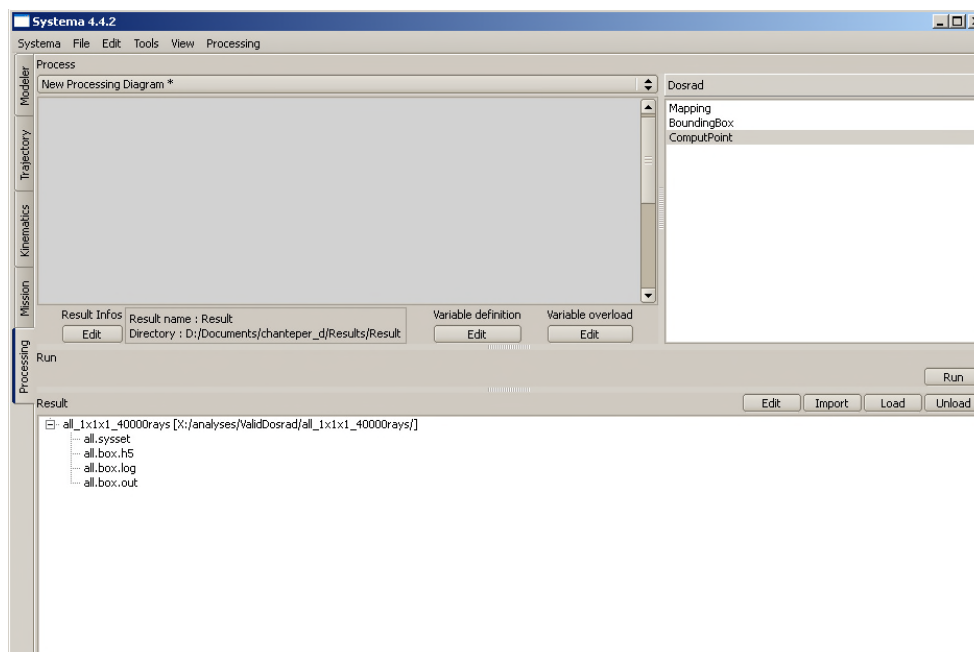


Refer to *SYSTEMA 4.4 User Guide (Chapter 8: “Process Management”, § “Using process result”)*.

The *.h5* files can be used to display the results in the **Modeler** tab.

1. Open the **Processing** tab in **SYSTEMA**.
2. Click **Load** to load the result file (*.sysres*).

Example

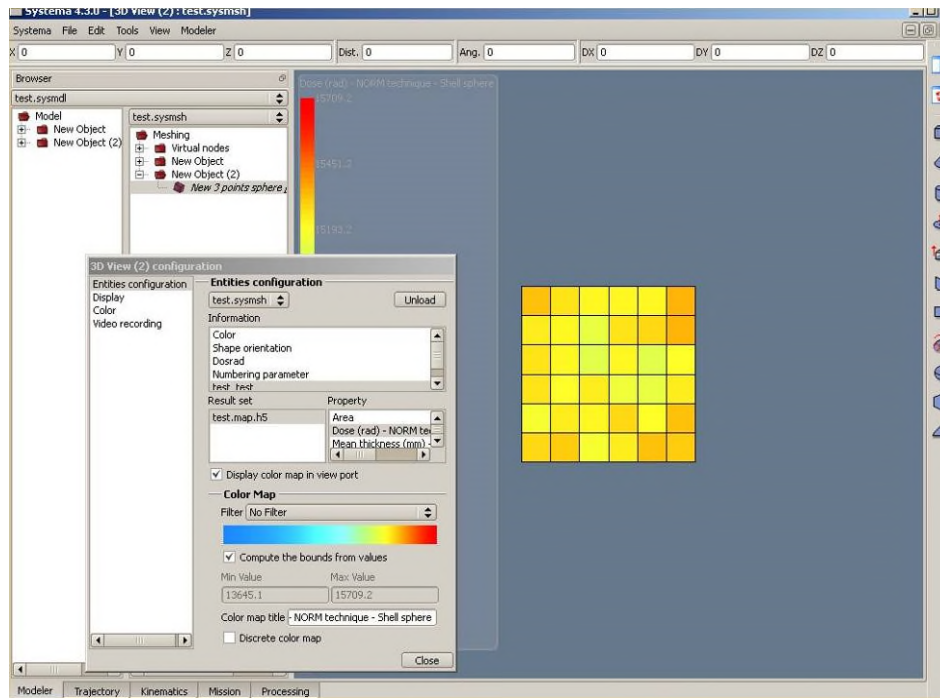


7.1 How to Display Results in a 3D View

Example

1. Open the **Modeler** tab in **SYSTEMA**.
2. Open the meshing model.
3. In the 3D view, via the right click menu, select **Configuration window**.
4. Select the result file and the property (for example: Dose) you wish to display.
5. Close the configuration window.

In the 3D view, the meshing model is displayed showing the result applied to the dose calculation.





Computation methods





8 General Method: Shielding Computation

8.1 Overview

The **DOSRAD** software evaluates the radiation dose received from space at a mesh or a virtual node. For the bounding box module, the point is surrounded by a shielding structure (represented by a virtual box).

The analysis is based on an analytical ray-tracing technique. Rays are traced in all directions from the point of interest (the technique depends of the computation module). Each ray explores the geometry in one direction. It passes through several objects and finally exits the model. A protection thickness and a density are associated with each shape of the model. The density is set with the selection of a material or can be modified.

All 3D and 2D objects are surface entities with no physical thickness (shell surfaces). In order to be able to treat the object, the user has to specify the "equivalent-thickness" for the study.

For each launched ray, the ray-tracing gives all the encountered surfaces of the geometrical model.

The dose received on the virtual nodes (bounding box or computation point) or on the meshes (dose cartography) is calculated as following:

- For each of launched rays, the amount of shielding encountered is determined, using the radiation geometrical model and taking into account the material density (in order to obtain the aluminium-equivalent thickness).

If one ray encounters no geometrical object but only the space, the user is informed on the mesh number where space was directly encountered (the information is given in the results file). This allows the user to verify its geometrical model and to consider the calculated dose with care. All objects give shielding against radiations except the objects linked to the Bounding Box and Computation Point specific items and the objects with no activity (see the application properties). Honeycomb and Volume specific items modify the shielding provided by their referenced surfaces.

This procedure is approximate, in that it ignores electron angular scattering and bremsstrahlung angular effects, but it is good for ion transport where attenuation is virtually in a straight line.

Depending on the main particles present in the environment, NORM or SLANT technique is used to determine the thickness crossed by a ray.

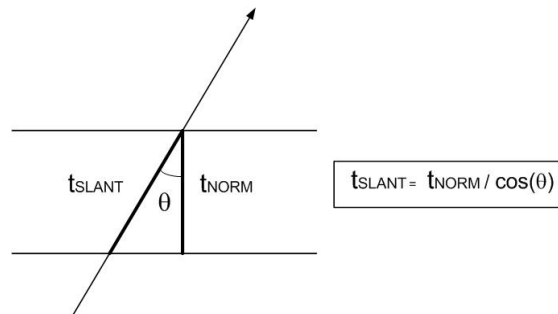


Figure 17 - Thickness computation: NORM and SLANT techniques

where:

t_{NORM} : Thickness defined in application properties in the geometrical model.

As demonstrated by Monte Carlo validations and analysis on ionising particles transport, there is a limited range of application for each of these techniques. Protons propagate in straight line while electrons propagate in an erratic way so it is recommended to compute proton shielding using SLANT technique and electron shielding using NORM technique.

An equivalent thickness for each ray is then calculated using NORM technique if shell sphere dose is used and SLANT technique if solid sphere dose is used. Monte Carlo comparisons lead to use the associations SLANT/SOLID SPHERE and NORM/SHELL SPHERE.

These two techniques lead to significantly different estimate of shielding thicknesses.

- From every aluminium-equivalent thickness on ray, an equivalent dose is calculated using the dose depth curve.

The dose curve is an input either automatically from the DOSE DEPTH CURVE application (only available in the V3 version) or directly by the user.

The dose computation interpolates in the dose depth curve the dose received in each direction depending on the encountered thickness. The computation of the dose is done by a semi-logarithmic interpolation. If the thickness is smaller than the thickness for the first point, the dose corresponding to this smallest thickness is taken. If the thickness is larger than the thickness for the last point, then the dose corresponding to this largest thickness is taken.

- The average dose per mesh (for the bounding box computation and dose cartography) or per computation point (for the computation point module) is then calculated considering the dose per ray launched in the considered mesh or computation point.

From dose per ray and the solid angle of meshes and faces (for the bounding box computation only), a dose per mesh and per face can be estimated. Each dose per face can give an equivalent thickness per face.



8.2 Honeycomb specific item

The **DOSRAD** software is able to take into account honeycomb panels that may composed a spacecraft.

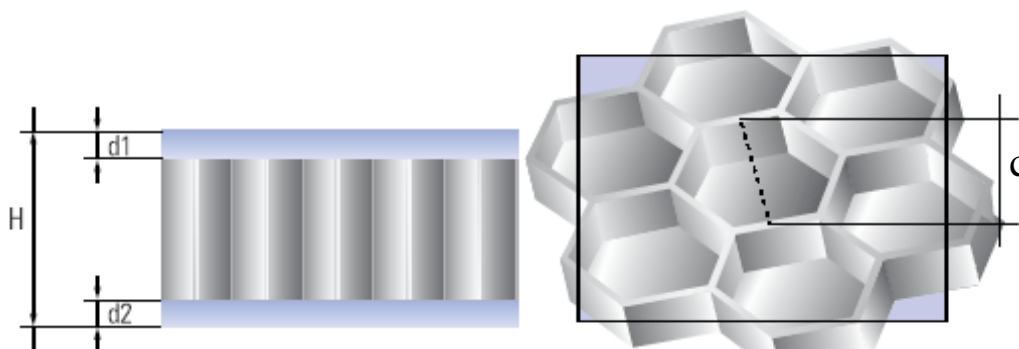
A honeycomb panel is composed of two face sheets and a core. The face sheets have to be modeled as standard shell shapes. Then the two face sheets have to be gathered in a specific item for the computation algorithm to account for the core.

The algorithm to compute the thickness encountered by a ray is modified as follows:

- If a ray encounter surfaces referenced in only one of the two face sheets of a honeycomb specific item, only the corresponding face sheet thickness is accounted for.
- If a ray encounter surfaces referenced in the two face sheets of a honeycomb specific item, the two corresponding face sheet thicknesses are accounted for. An extra thickness is added to model the presence of all the foils composing the honeycomb core (see Figure 2).

The added extra thickness is dependent on the honeycomb panel properties: nominal panel thickness, cell size, foil thickness and density. It depends also on the ray impacting angle with respect to the surface normal. The dependence of this additional thickness with the impacting angle is based on geometrical consideration.

The honeycomb core properties are:



	Unit	Example
Nominal Panel Thickness H	m	0.02015
Foil thickness	1/10000 inches	10
Cell size c	Inches	3/16
density (of the foil)	kg/m ³	2640

Figure 18 - Characteristic honeycomb core properties

The user has to input the sheet's thickness and density on the geometrical model whereas the honeycomb core properties have to be input in the honeycomb specific item (meshing).



8.3 Volume specific item

Usually in ray-tracing radiation analysis tool, spacecraft and/or hardware are geometrically represented by shell surfaces, called shapes in **SYSTEMA**. In some cases, it can be useful for the user to represent some parts of the model by a volume rather than by a shape. Since version 4.5.0, this is possible, either when using the SLANT or the NORM technique for thickness computation.

A volume in the **DOSRAD** software is defined by the set of its limiting shapes and a density. As such, this is a specific item.

The algorithm to compute the SLANT thickness encountered by a ray is modified as follows:

- If a ray encounters surfaces referenced in a volume, the number of these surfaces determines if the ray originates from inside the volume (odd number of surfaces) or from outside (even number of surfaces).
- The SLANT thickness is computed as the sum of distances between entering and exiting points.

Of course, the thickness set as **DOSRAD** properties in the model part are not accounted for anymore for the volume's referenced shapes.



The fact that the shapes referenced by a volume are effectively representing the border of a volume is not automatically checked. The user shall thus verify this by visual inspection (no hole).



It is possible to perform bounding box analysis on a volume. To do so, the surfaces referenced by the volume must coincide with the surfaces referenced by the bounding box. In this case, the thicknesses generated by the ray / referenced volume interaction will not be taken into account.

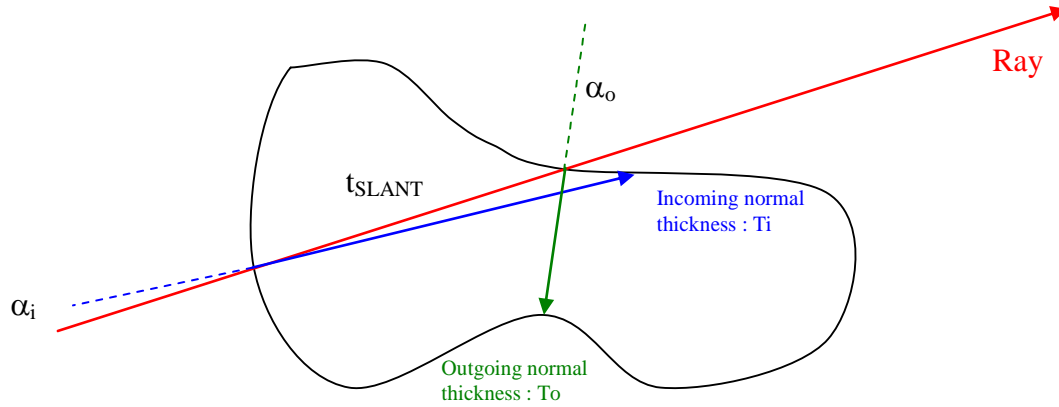
The algorithm to compute the NORM thickness encountered by a ray is modified as follows:

- If a ray encounters surfaces referenced in a volume, the number of these surfaces determines if the ray originates from inside the volume (odd number of surfaces) or from outside (even number of surfaces).
- The NORM thickness is computed as the sum of estimated normal distances. These minimal distances depend on the user choice concerning the method to be used, see figure 3.

Note that the last method (Cosine Weighted) does not lead to an increase in computation time, whereas the other three necessitate generating supplementary rays that may cause an increase in computation time.



The choice of the NORM method is available in the Ray Tracing tab of each processing settings corresponding to the three modules.



	Method	Thickness
1 (default)	Incoming/outgoing Min Path	$\text{MIN}(t_{SLANT}, T_i, T_o)$
2	Incoming Min Path	$\text{MIN}(t_{SLANT}, T_i)$
3	Weighted Min Path	$\text{MIN}[t_{SLANT}, 0.5*(T_i+T_o)]$
4	Cosine Weighted	$t_{SLANT} * 0.5 * (\cos \alpha_i + \cos \alpha_o)$

Figure 19 - Computed NORM thickness in the case of impact with a volume



9 Dose Cartography

9.1 Theory

Introduction

The dose cartography is performed at spacecraft or equipment level for the calculation of the total dose received on model meshes. It gives dose mapping on surface whatever their shape (cone, cylinder, sphere...). It enables to get a rapid estimation of dose levels expected on objects (for mirrors, lenses for example) or to think about the satellite agencement in term of radiation impact.

For all surfaces that are meshed (i.e. spacecraft surfaces and not virtual surfaces), a dose mapping is computed. From each mesh, a certain number of rays are launched. Both the emission point, i.e. the point in the mesh from which the ray is fired, and the emission direction are sampled randomly. From all rays in a mesh, a mean total dose is calculated.

Monte-Carlo ray tracing method

This is the best method for an accurate modelling. The general principle is:

- Simulation of the particles behavior by considering the individual history of a small number of particles.
- The particles are sampled randomly from the appropriate emission distribution function.
- The macroscopic behavior is reached statistically after firing a sufficient number of particles.

The computation of the paths followed by the rays within the physical model is called "ray tracing". The ray tracing method takes the Lagrangian point of view by considering the individual history of particles from a point of emission. The rays are launched from the surfaces of the spacecraft to space, to go back in history of particles.

Consider the energy emitted from a mesh "m" per unit time to be subdivided into N_m individual packets. It is assumed that the number of energy packets $d N_m$ emitted from any surface element $d A_m$ will be in the proportion of the total number of energy packets emitted by the mesh "m":

$$\frac{d N_m}{N_m} = \frac{d A_m}{A_m} \quad (\text{Equation 1})$$



The Monte-Carlo ray tracing method includes the following modelling tasks:

- Emission of rays from a mesh (only one ray is fired per surface point).
 - Emission points are sampled randomly, according to an iso-distribution law, with an analytic definition of shapes.
 - Emission directions are sampled randomly, according to an isotropic distribution law over 4π sr.
- Propagation of rays and search for impacts.
 - The main problem is to determine the surface hit by the ray.
 - An optimized search is needed as it is impossible to test all the surfaces.
 - The ray tracing considers the analytic definition of shapes.

Surface sampling

Let β and γ be the parameters of the surface.

It is assumed that any mesh of the surface is associated with a rectangle in the parametric space, defined by its limits β_1, β_2 and γ_1, γ_2 .

The procedure for sampling randomly a point belonging to a mesh is equivalent to the sampling of two random values for β and γ , verifying the above-displayed equation 1.

The method for sampling randomly β and γ is presented here. Except for the quadrangle, the infinitesimal area dA of a surface can be written as the product of two separate functions of β and γ :

$$dA = F(\beta)d\beta \cdot G(\gamma)d\gamma$$

the equation 1 becomes:

$$\frac{dA}{\int_{\beta_1}^{\beta_2} F(\beta) d\beta \cdot \int_{\gamma_1}^{\gamma_2} G(\gamma) d\gamma} = \frac{F(\beta)d\beta}{\int_{\beta_1}^{\beta_2} F(\beta) d\beta} \cdot \frac{G(\gamma)d\gamma}{\int_{\gamma_1}^{\gamma_2} G(\gamma) d\gamma}$$

It clearly indicates that when normalized, F and G represent the distribution functions of the random variables β and γ .

The computer random generator returns equi-probable numbers valued in [0,1]. Let R be such a random number. The value of β is obtained by solving the following equation:

$$R = \frac{\int_{\beta_1}^{\beta} F(t) dt}{\int_{\beta_1}^{\beta_2} F(t) dt}$$



For the quadrangle, it is not possible to separate β and γ and the previous technique is not easy to implement.

An alternative technique is to consider rays with a variable energy. Consider again the N individual rays (packets of energy) emitted from a mesh, each of different energy, dE_k , the above-displayed equation 1 becomes:

$$\frac{dE_k}{E} = \frac{dA_k}{A} \quad (\text{Equation 2})$$

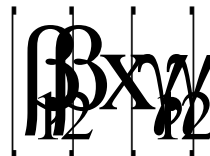
$$1 \leq k \leq N$$

if you take $dA_k = H(\beta_k, \gamma_k) d\beta d\gamma$, the above equation 2 becomes:

$$\frac{dE_k}{E} = \frac{H(\beta_k, \gamma_k) d\beta d\gamma}{A} \quad (\text{Equation 3})$$

dE_k appears to be proportional to $H(\beta_k, \gamma_k)$.

Thus, a “statistical weight” proportional to $H(\beta, \gamma)$ is multiplied to the energy of each ray emitted from a point in the quadrangle, whose parametric coordinates β and γ are sampled equi-probably and randomly in



Ray direction sampling

The ray direction is sampled randomly, based on the distribution function corresponding to an isotropic law over 4π sr. Starting from two machine random numbers R_1 and R_2 , the angles θ (angle with the surface normal vector) and ψ (angle around the normal vector) are computed by:

$$\psi = 2\pi R_1$$

$$\theta = \arcsin(2R_2 - 1)$$

Computation algorithm

The number of rays that are launched from each mesh of the model is determined by the user choice. The user can specify a density of rays or a given number of rays.

For each ray of a mesh, the dose is computed as follows:



- Random choice of the emission point and ray direction.
- Ray-tracing method (propagation of the ray and intersection with surfaces).
- Equivalent-aluminium thickness computation (two techniques are used NORM and SLANT, it depends on the type of the dose curve).
- Dose computation by interpolating the dose depth curve.

From all rays of the considered mesh, the average dose is computed and then the equivalent-aluminium thickness (by interpolating the dose depth curve).

The algorithm also includes:

- The mean thickness computation, for which the dose depth curve is not used.
- The number of rays that encountered no objects in the model ("void rays") and the number of rays that encountered at least one object with a null thickness ("thick=0 rays").



The surface on which the calculation is performed is not taken into account in the calculation, which means that it does not give any shielding for thickness computation. If an object or a complex shape, i.e. made of several shapes such as a box, is meshed, only the surface on which the emission point is located gives no shielding for its computation, whereas the other surfaces (such as the five other faces of a box) give shielding to radiation.

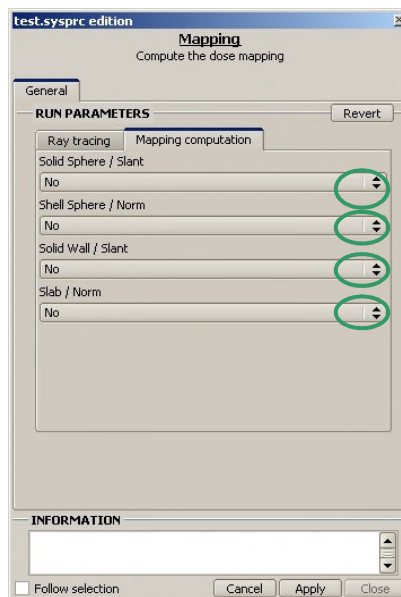
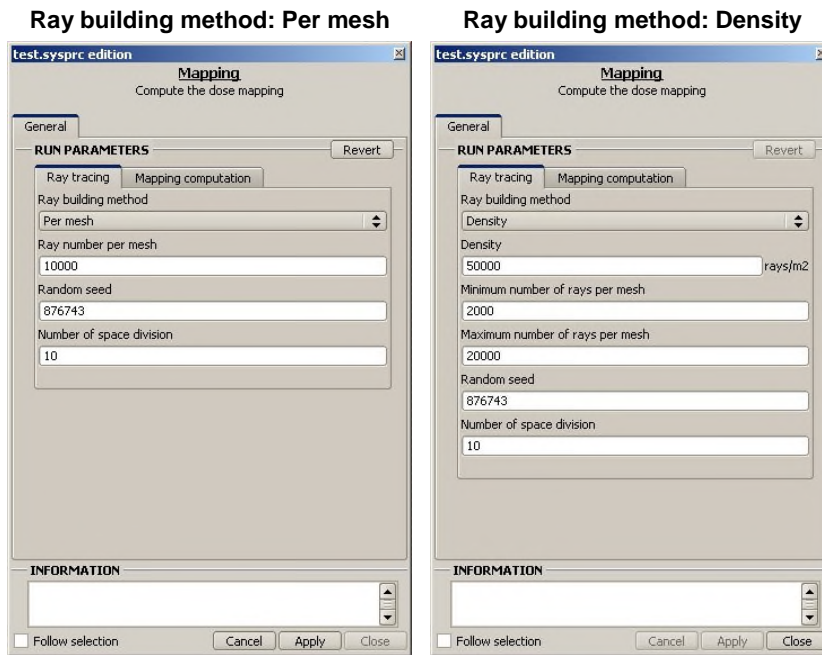


9.2 Parameters

This section gives an overview of the windows in this module.



The parameters set in the different fields correspond to the default values.



These drop-down lists let the user select or not a dose calculation technique for the mapping



The user must at least select a method in accordance with the dose curve data.

9.3 Input / Output



In this **DOSRAD** User Manual, refer to the General Principles in Chapter 2 (DOSRAD Modules) for the "INPUT" and "OUTPUT" tables of the different modules and corresponding files.

Input

- .SYSSET file
 - **Mission synthesis file for the DOSRAD application.**
- .DOSE file
 - **Description file of the dose depth curve.**

Output

- MAP.LOG file
 - **General information file.**
- MAP.H5 file, RMAP.H5 file
 - **Used to display the doses, the thicknesses and possibly the rays that have been fired colored by encountered thicknesses.**



10 Bounding Box Analysis

10.1 Theory

Introduction

The bounding box analysis is performed at system level for the computation of the six aluminium- equivalent thickness of any equipment for subcontractor specification. These six thicknesses represent the shielding given by the satellite to the analysed equipment (seen from the computation point). This two-step analysis (system level followed by equipment level analysis) allows to handle with less complex model without large data exchanges (between system and equipment description). If six aluminium-equivalent thicknesses are defined with adequate margins, the equipment level radiation analysis may be re-used for all satellite programs.

Bounding box definition

The bounding box is defined in the meshing model from one or several shapes of the geometrical model. If only one object is selected, the bounding box name is given by the selected object followed by "bounding box". By default, the bounding box is defined as the smallest box with axes parallel to the system frame including these shapes. The default meshing is 5x5 for all faces. Orientation, size, meshing and referenced shapes may be changed.

One computation point is systematically added to this bounding box. By default, at the bounding box creation, the attached computation point is set to the bounding box center. By default, bounding box referenced shapes are also computation point referenced shapes. These attached shapes give no shielding to radiation in the bounding box analysis. Computation point position and referenced shapes may be changed.

Computation algorithm

The analysis is performed from the computation point of the bounding box. A solid angle sectoring technique is used to obtain the dose received at the point of interest.

The number of rays that are launched from the computation point through each mesh of the bounding box is determined by the user choice. The user can specify a surface density or a given number of rays.

For each mesh of the considered bounding box, the analysis is performed as follows:

- The number of rays in the two dimensions of the mesh can be computed.
- Computation of the solid angle between the computation point and the mesh. The solid angle per face is also updated.
- For each ray in the two dimensions of the mesh, the dose is computed as follows:
 - **Discretization of the mesh: a point is computed (regular distribution of the points).**



- **Emission direction (from the computation point to the point of discretization).**
- **Computation of the elemental solid angle around the ray.**
- **Ray-tracing method (propagation of the ray and intersection with surfaces).**
- **Equivalent-aluminium thickness computation (two techniques are used (NORM and SLANT) depending on the type of the dose curve. If M objects are encountered along the ray, the thickness t_i is as following:**

$$t_i = \sum_{j=1}^M t_j \frac{d_j}{2,7}$$

where t_j is the j object thickness (NORM or SLANT) and d_j is the j object density (expressed in g/cm³ in the above equation).

- **Dose computation by interpolating the dose depth curve.**
 - **Update of the dose per mesh.**
 - **Mean thickness computation (the dose depth curve is not used).**
 - **Computation of the number of rays that encountered no object in the model (“void rays”) and the number of rays that encountered at least on object with a null thickness (“thick=0 rays”).**
- Computation of the thickness for the considered mesh.
 - Dose per face can be estimated.

Each dose per face gives an equivalent-aluminium thickness per face. The total dose received at the computation point is then approximated as:

$$D = \sum_{i=1}^n d(t_i) \frac{\Omega_i}{4\pi}$$

where:

d(t_i) : Dose at the centre of a t_i radius aluminium sphere when solid sphere dose curve is used.
 Dose at the centre of a t_i shell thickness when a shell dose curve is used

Ω_i : Elemental solid angle around ray i. The sum is over all elemental solid angles.

The equivalent total thickness can also be computed.

When calculation is made using the solid sphere dose curve and the SLANT technique, an automatic calculation is run to evaluate the equivalent-aluminium thicknesses that



must be transferred to the subcontractors working with the same method and dose curve in order that they **get the same total dose on the computation point of their equipment**. The method is based on dichotomy. The real thicknesses per face to be transmitted to subcontractors, called SLANT FOR SUB, are always lower than the SLANT thicknesses calculated during the analysis. These thicknesses are deduced from the dose per face:

$$T_{face} = d^{-1} \left(4\pi D_{face} / \Omega_{face} \right)$$

Discretization of a bounding box mesh

Points corresponding to each ray are equi-distributed on the bounding box meshes. The directions of these rays are defined by the line from the computation point position and passing through the points on the meshes. The result for the mesh is the average of all the results for all rays.

Figure 20 - Rays are launched with a regular angular distribution



10.2 Parameters

This section gives an overview of the windows in this module.



The parameters set in the different fields correspond to the default values.

Ray building method: Per mesh

New Processing Diagram edition
BoundingBox
 Compute the dose on bounding boxes

General

RUN PARAMETERS Revert

Ray tracing | Bounding Box computation | Bounding Box outputs

Ray building method: Per mesh

Ray number per mesh: 400

INFORMATION

Follow selection Cancel Apply Close

Ray building method: Density

New Processing Diagram edition
BoundingBox
 Compute the dose on bounding boxes

General

RUN PARAMETERS Revert

Ray tracing | Bounding Box computation | Bounding Box outputs

Ray building method: Density

Density: 50 rays/m2

Minimum number of rays per mesh: 10

Maximum number of rays per mesh: 15

INFORMATION

Follow selection Cancel Apply Close

New Processing Diagram edition
BoudingBox
 Compute the dose on bounding boxes

General

RUN PARAMETERS Revert

Ray tracing | Bouding Box computation | Bouding Box outputs

Solid Sphere / Slant: Yes

Shell Sphere / Norm: Yes

Solid Wall / Slant: No

Slab / Norm: No

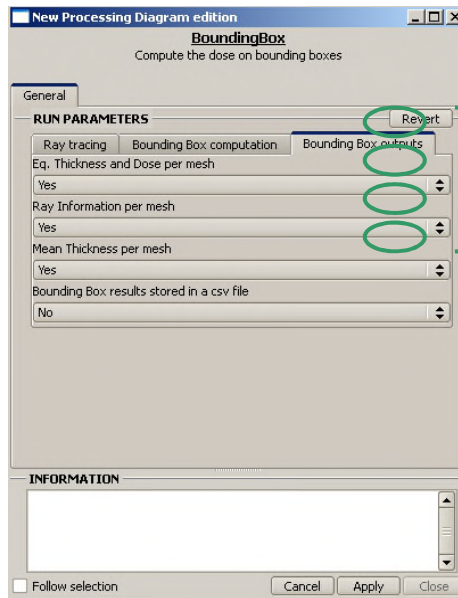
INFORMATION

Follow selection Cancel Apply Close

These drop-down lists let the user select or not a dose calculation technique on bounding boxes



The user must at least select a method in accordance with the dose curve data.



These drop-down lists let the user select or not a results detailed in results file. The last one allows to activate an extra output file (under csv format) containing faces information results

10.3 Input / Output



In this **DOSRAD** User Manual, refer to the General Principles in Chapter 2 (**DOSRAD** Modules) for the "INPUT" and "OUTPUT" tables of the different modules and corresponding files.

Input

- .SYSSET file
 - **Mission synthesis file for the **DOSRAD** application.**
- .DOSE file
 - **Description file of the dose depth curve.**

Output

- BOX.LOG file
 - **General information file.**
- BOX.H5 file, RBOX.H5 file
 - **Description file of dose calculation results on bounding boxes. Used to display the doses, the thicknesses and possibly the rays that have been fired colored by encountered thicknesses.**



- BOX.OUT file
 - **Bounding boxes output file as ASCII format.**
- BOX.CSV file
 - **Optional Bounding boxes output file as ASCII / CSV format.**



11 Computation Point Analysis

11.1 Theory

Introduction

The computation point analysis is performed at system or equipment level for the estimation of the total dose at one point in the geometrical model. It uses the Monte-Carlo ray tracing method. A certain number of rays are launched from the computation point. The emission direction is sampled randomly. From all rays, a mean total dose is calculated.

The Monte-Carlo ray tracing method is explained in the “Dose cartography” chapter. The difference is that here, the emission points are already set by the user (no surface sampling).

Computation point definition

The computation point is defined in the meshing model by its position and referenced shapes. At its creation, if only one object is selected, the computation point name is given by the selected object followed by "computation point".

By default, its position is set to the system frame origin. There may be zero, one or several referenced shapes.

These referenced shapes give no shielding to radiation in the radiation analysis. Computation point position and referenced shapes may be changed.

Computation algorithm

The number of rays that are launched from the computation point is specified by the user.

For each ray, the dose is computed as follows:

- Random choice of the ray direction. The ray is launched with a randomly isotropic distribution over 4π sr (refer to “Ray direction sampling” in “Dose Cartography” section 2).
- Ray tracing method (propagation of the ray and intersection with surfaces).
- Equivalent-aluminium thickness computation (with NORM and SLANT techniques) from all objects encountered along this ray (the method is the same used for the dose cartography and the bounding box analysis).
- Dose computation using the dose depth curve.



From all ray contributions, the average total dose is obtained and then the equivalent-aluminium thickness.

The algorithm also includes:

- The mean thickness computation, for which the dose depth curve is not used.
- The number of rays that encountered no objects in the model (“void rays”) and the number of rays that encountered at least one object with a null thickness (“thick=0 rays”).

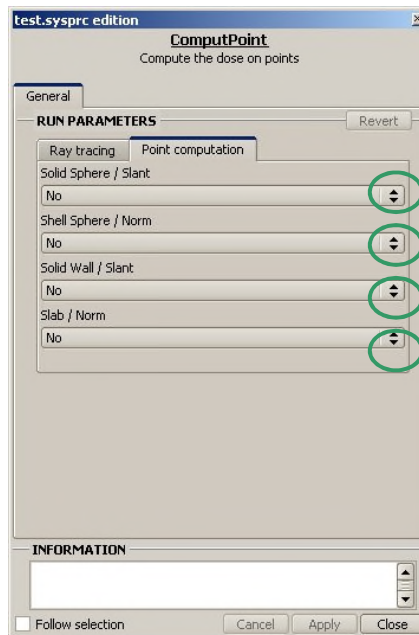
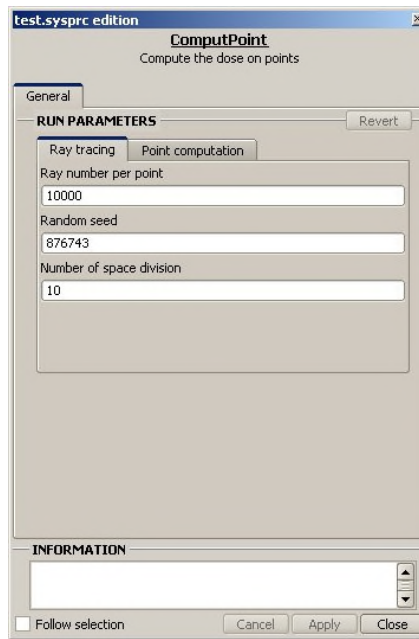


11.2 Parameters

This section gives an overview of the windows in this module.



The parameters set in the different fields correspond to the default values.



These drop-down lists let the user select or not a dose calculation technique on computation points



The user must at least select a method in accordance with the dose curve data.

11.3 Input /Output



In this **DOSRAD** User Manual, refer to the General Principles in Chapter 2 (DOSRAD Modules) for the "INPUT" and "OUTPUT" tables of the different modules and corresponding files.

Input

- .SYSSET file
 - **Mission synthesis file for the DOSRAD application.**
- .DOSE file
 - **Description file of the dose depth curve.**

Output

- PT.LOG file
 - **General information file.**
- PT.OUT file
 - **Computation Point output file as ASCII format.**
- PT.H5 file, RPT.H5 file
 - **Description file of dose calculation results at computation points. Used to display the doses, the thicknesses and possibly colored by encountered thicknesses.**



Getting Started with Dosrad



12 Introduction

12.1 Overview

Welcome to **DOSRAD** Getting Started level course. In this course, you will learn how to perform a radiation analysis using the **DOSRAD** application.

The Getting Started describes the complete process to perform this analysis.

It includes the following parts:

- In the “DosradSat” folder:

<i>Material definitions for bulk</i>	<i>(sysmtr file)</i>
<i>Complete geometrical model with application properties</i>	<i>(sysmdl file)</i>
<i>Meshing & numbering associated with the geometry</i>	<i>(sysmsh files)</i>
<i>Mission gathering all previous elements</i>	<i>(sysmis files)</i>
<i>Dose curve file</i>	<i>(dose file)</i>
<i>Process diagram already set to perform the DOSRAD modules</i>	<i>(sysprc files)</i>

In addition to those files, a sub-folder “Results” contains previously computed results. If you run the included process, the results will be created in a folder “My_Results” within the related SYSTEMA_WORKING_DIR folder.

These files should not be modified and must be used as references to help completing the “MySat” model.

- In the “MySat” folder:

<i>Material definitions for bulk</i>	<i>(sysmtr file)</i>
<i>Geometrical model without application properties</i>	<i>(sysmdl file)</i>
<i>Dose curve file</i>	<i>(dose file)</i>

This folder only includes the basic inputs to start working with **DOSRAD**.

The creation of those kinds of files is explained in the Systema v4 Manual and does not depend on the **DOSRAD** application itself. This is why they are considered as inputs in this Getting Started.



*Even if the creation of the geometry itself does not depend specifically on **DOSRAD**, the design of the geometry must comply with the expected radiation dose modeling.*



12.2 The DosradSAT Model

The DosradSat model is a simplified satellite designed for a simple radiation analysis at system level.

It is composed of:

- **A platform made of HC (honey-comb) walls covered by MLI**

This structure has been modeled using two thicknesses for the two materials.

- **Equipment**

Equipment is made of a surrounding aluminum box. As regards the inner part of the equipment, it is represented by a sphere.

- **Radiators**
- **A functional instrument**

The instrument presented here is modeled only with inactive shapes. It does not participate in the radiation simulation performed here.

However, some parts of the instrument have been kept active to evaluate the fluxes collected by those geometries.

- **Solar panels**

The solar panels are made of sun cells.



12.3 In this Chapter

In a first stage, you will explore the **SYSTEMA** platform and its functionalities. For that purpose, you will perform the following:

- **Open an existing model.**
- **Explore the pre-processing options.**
- **Load the results.**
- **Perform post-processing operations.**

In a second stage, you will learn how to perform the radiation dose analysis described in this Getting Started. To do so, you will have to follow these steps:

- **Set the material references.**
- **Complete the geometrical model with the applicative properties.**
- **Create the meshing.**
- **Create the mission.**
- **Create a process diagram and run the process.**



For the preliminary steps (creation of geometry, material file, meshing, mission), refer to the Systema Getting Started Manual.



13 DosradSat

13.1 Introduction

In this chapter you will start working from an existing model to get familiar with the DosradSat environment.

With the DosradSat model delivered with the **DOSRAD** software application, you will be able to pre-process and post-process data and loading existing results.

In the sections below you will navigate through **SYSTEMA** and identify the main functions of **DOSRAD**.

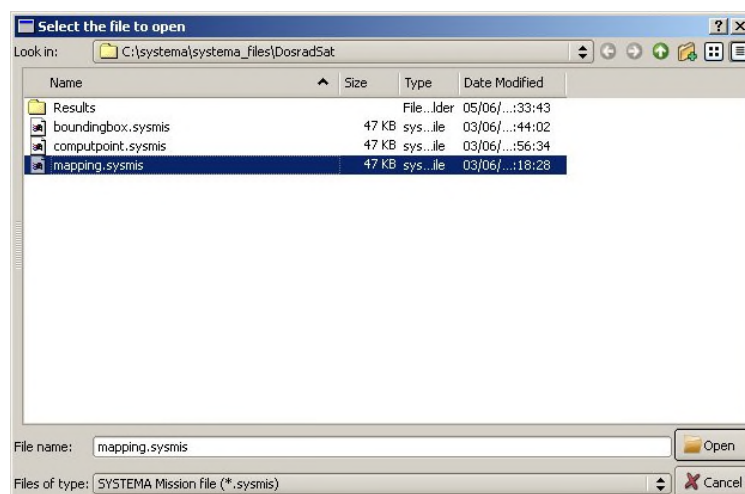
13.2 Opening an Existing Model

Using a DosradSat model

To open the existing DosradSat model:

1. Open the **Mission** tab in **SYSTEMA**.
2. Click the **File** -> **Open** menu.

The **Select the file to open** dialog box opens:



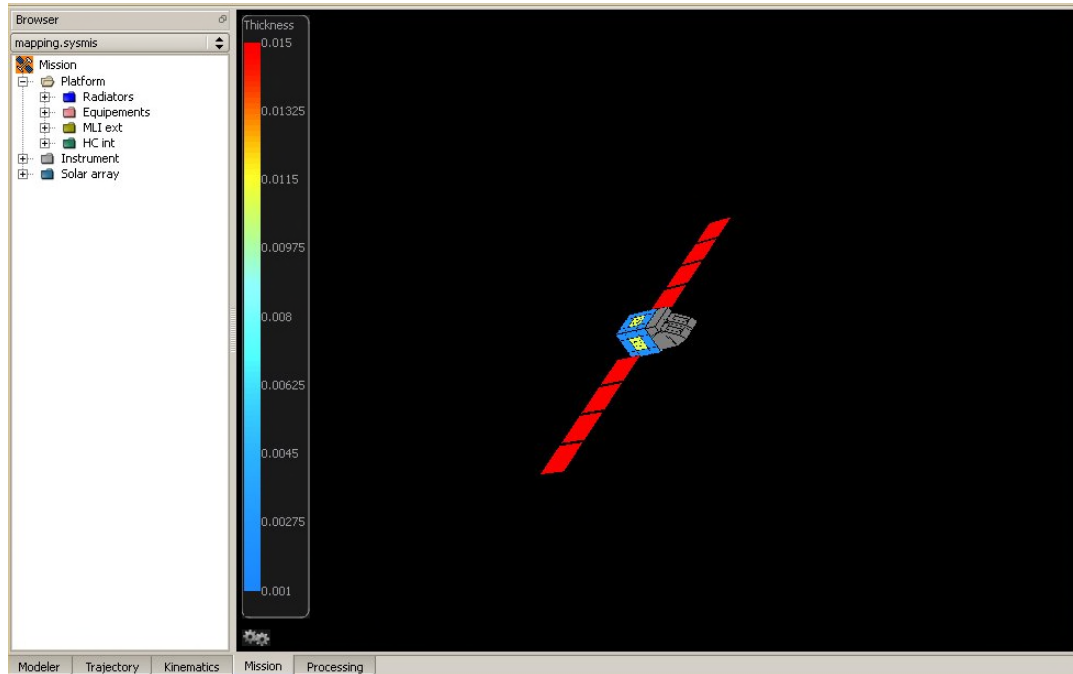
3. Open the *.sysmis file (*mapping.sysmis* file, for example).



When you open a file with the Mission tab, all the associated files (geometrical model, meshing) open.



All the elements of the satellite are graphically represented:





13.3 Pre-Processing

After the model has been loaded, you may want to pre-process the results (e.g. check the properties set for the model).

To open the configuration panel:

- Right-click the view and select **Configuration window**.

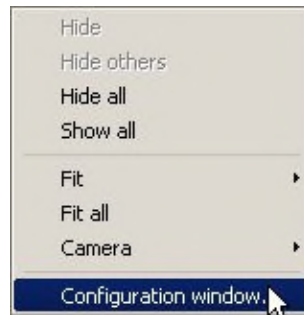



Figure 21 - Pop-up menu

OR

- Click  in the left corner of the view.

The **3D View Configuration** window opens:

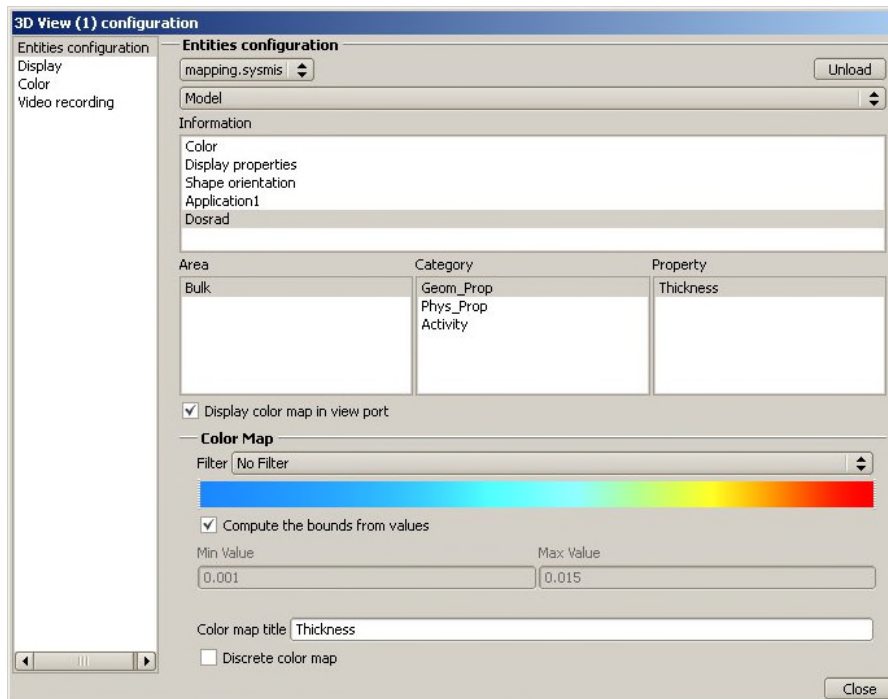




Figure 22 - 3D view window

The data that can be viewed at this stage are as follows:

- **COLOR:**

Refers to the colors defined in the model.

- **Shape orientation:**

Refers to the orientation of faces (the red color is used for positive faces and the blue color is used for negative faces).

- **Dosrad:**

Refers to the **DOSRAD** properties (with the Area, Category and Property containers).



The applicative properties defined in the mission are automatically displayed at this stage.



The grey color in the view means that the property has not been set.



13.4 Loading Results

Loading results related to DosradSat

You can load results from the result area in the **Processing** tab.

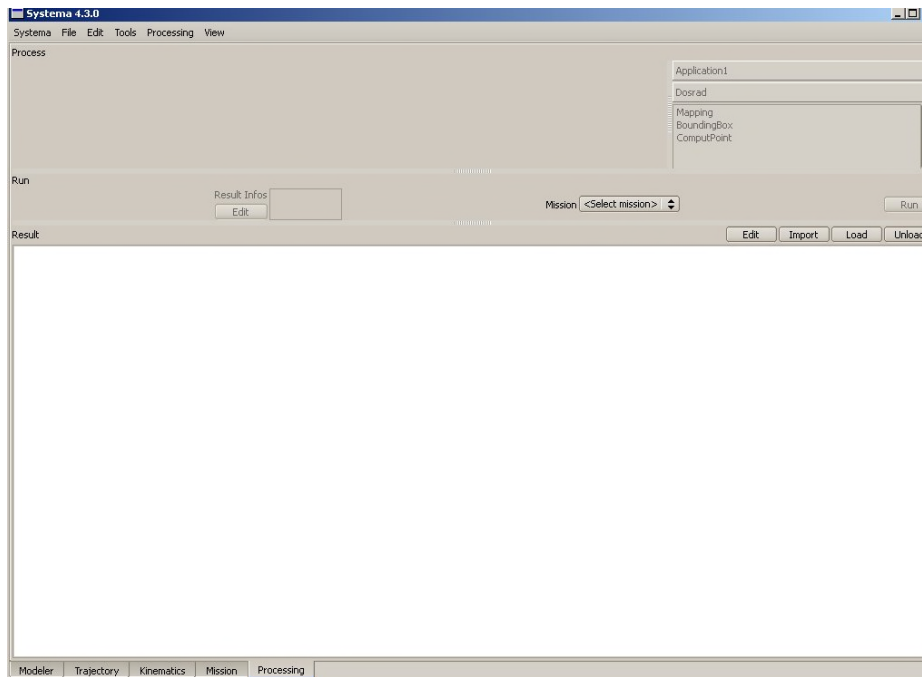


Figure 23 - Result area

This area includes four buttons with the following functions:

- **Edit**
Used to edit a text file.
- **Import**
Used to import any result file (e.g. an *hf5* file).
- **Load**
Used to download a set of results created upon a diagram execution.
- **Unload**
Used to unload the open files.




These results are available in the Modeler tab.

13.5 Post-Processing



In the **Modeler** tab, you will be able to display results on the meshing

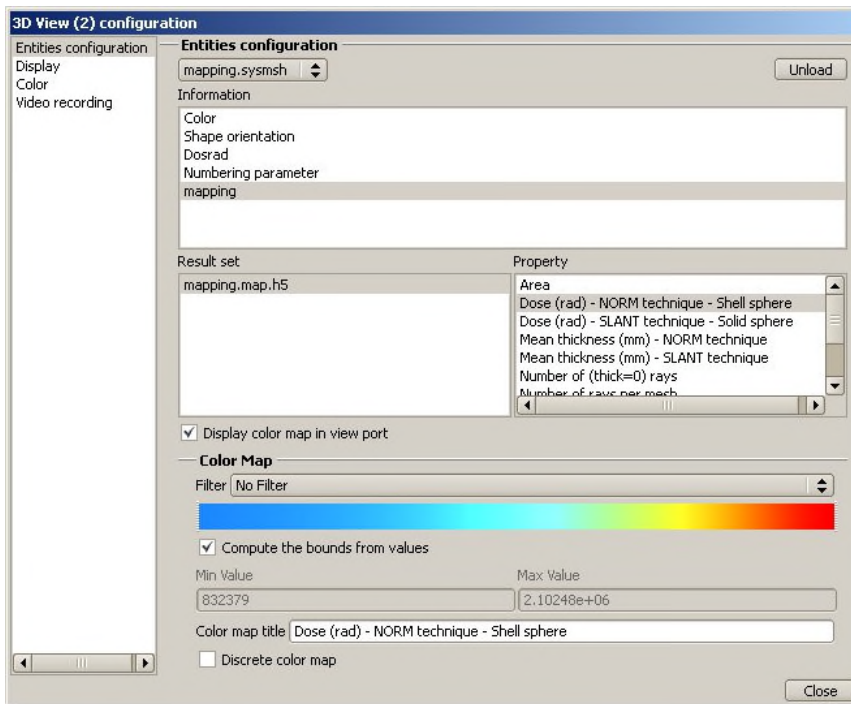
1. Open the **3D View Configuration** window by clicking .
2. According to the results (mapping, boundingbox or computpoint) you want to load:

Click **mapping**, **boundingbox** or **computpoint** in the Information area list.

The *.h5* files used to pre-process the data are displayed.

If you select the *mapping.map.h5* file, all the elements contained in the mapping results display (**Property** list box).

Example



3. Click **Close**.

Here you can, for instance, display mapping results.

Example

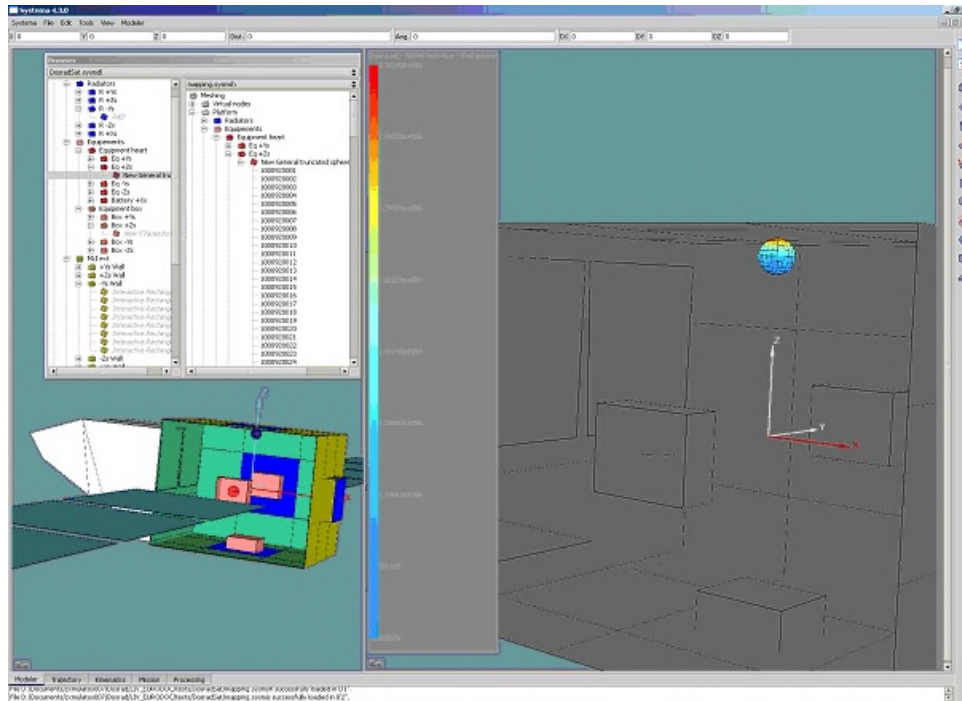


Figure 24 - 3D view



14 MySat

14.1 Introduction

Now you are familiar with the different functionalities of the **DOSRAD** environment, let's see how to create your own model.

To perform the radiation analysis, you must use the "MySat" model.

The model is built in accordance with what you want to model.



*The "MySat" model is the model you will create during this tutorial. The "DosradSat" model is delivered with **DOSRAD** and can be used for result comparison.*

Starting with MySat

You will start from the geometrical model files including the raw data of **SYSTEMA**.

To get access to the model:

1. Start the **SYSTEMA** application.
2. Click the **Modeler** tab in the lower part of the main window.

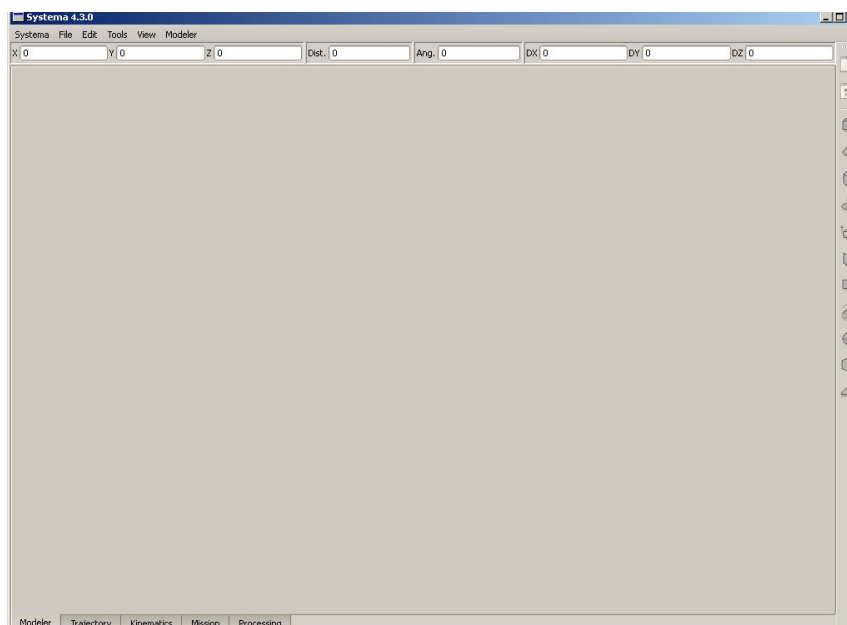
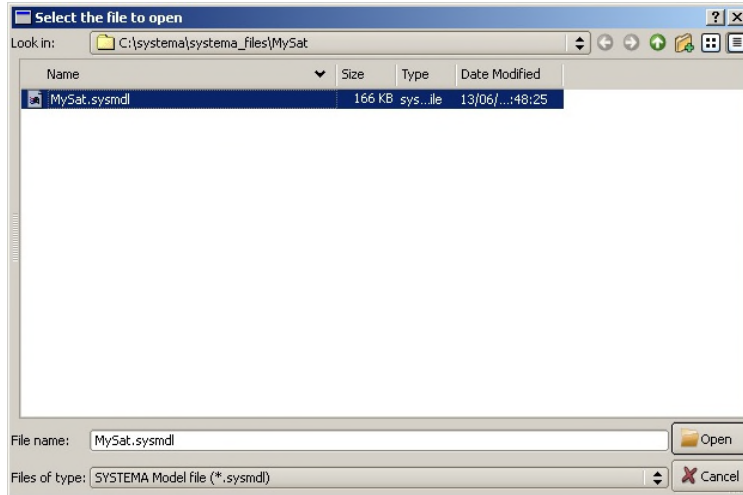


Figure 25 - Modeler tab



3. Click **File -> Open -> MySat**.
4. Open the *MySat.sysmdl* file.



All the shapes are displayed on an orange background in the model browser:

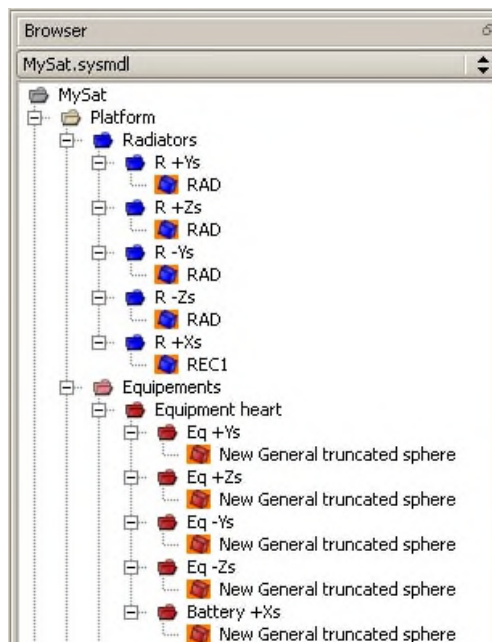
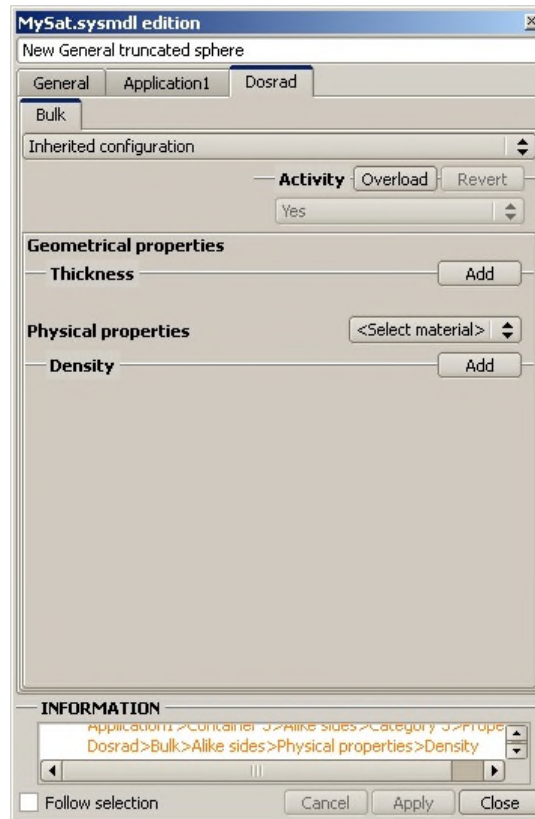


Figure 26 - Model tree structure



If you double-click a shape on orange background, a warning displays in the Information area.



14.2 Setting the Material References

The applicative properties will be defined from the material file. A default material database is embedded within Dosrad, but in this example we assume the user wants to attach its own.

To set the material references:

1. In the tree structure, double-click the **MySat** root directory.

The **MySat sysmdl edition** window opens:

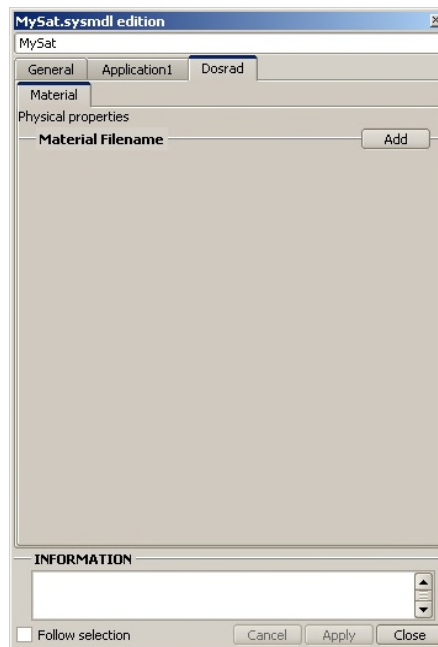
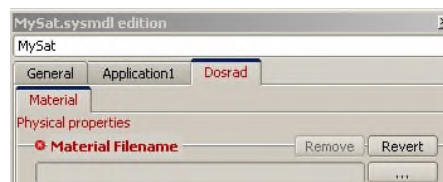



Figure 27 - Edit window

2. In the **DOSRAD** tab, click **Add** to open the Material Filename area.



3. In the Material Filename area, click  to select the material file.
4. Select the *DosradBulk.sysmtr* file.
5. Click **Apply**.



Now you will be able to enter the properties using the selected material file.

14.3 Completing the Geometrical Model

The general procedure is as follows:

- Set all the elements to “inherited configuration”.
- The highest objects will be set to “local configuration”.

Example

To enter the properties of the radiators:

1. In the Models browser, double-click **Radiators**.

The edition window dedicated to radiators opens:

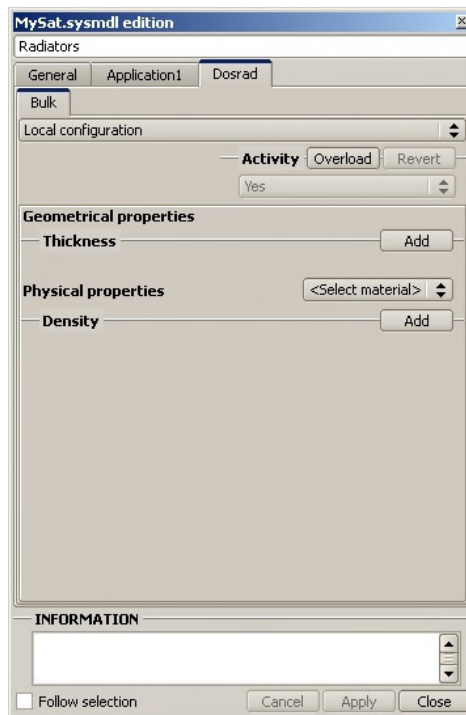


Figure 28 - Radiators edit window

2. Enter the **Bulk** properties.



At this stage, set the elements to Local configuration. The next elements will be set to Inherited configuration.



3. Enter the properties for each element.



To enter the properties, refer to the values in the tables below.

The elements to be set are as follows:

○ Radiators

Bulk	Configuration	Local configuration
	Activity	Yes
	Geometrical properties	
	- Thickness	0.011
	Physical properties	Radiator

○ Equipment heart

Bulk	Configuration	Local configuration
	Activity	No
	Geometrical properties	
	- Thickness	0.001
	Physical properties	Select "Aluminium"

○ Equipment box

Bulk	Configuration	Local configuration
	Activity	Yes
	Geometrical properties	
	- Thickness	0.003
	Physical properties	Aluminium



○ **MLI ext**

Bulk	Configuration	Local configuration
	Activity	Yes
	Geometrical properties	
	- Thickness	0.002
	Physical properties	MLI

○ **HC int**

Bulk	Configuration	Local configuration
	Activity	Yes
	Geometrical properties	
	- Thickness	0.006
	Physical properties	HC

○ **Instrument**

Bulk	Configuration	Local configuration
	Activity	No
	Geometrical properties	
	- Thickness	0.001
	Physical properties	Aluminium

○ **Optical (below “Ins -Y” and “Ins +Y”)**

Bulk	Configuration	Local configuration
	Activity	Yes
	Geometrical properties	
	- Thickness	0.009
	Physical properties	Optic



○ **Solar array**

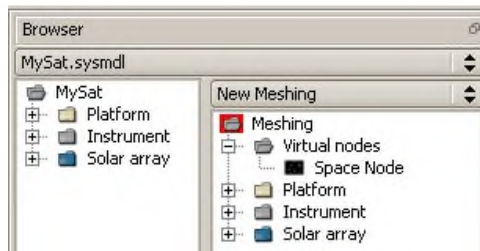
Bulk	Configuration	Local configuration
	Activity	Yes
	Geometrical properties	
	- Thickness	0.015
	Physical properties	<select material>
	- Density	4832

14.4 Creating the Meshing

Initializing the meshing file

To initialize a meshing, perform the following steps:

1. Click the **File -> Meshing -> New** menu.



2. In the browser, double-click **Meshing**.

The following window opens:

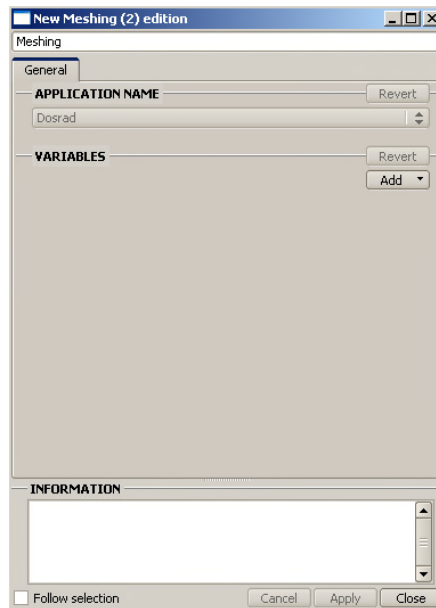


Figure 29 - New **Meshing** edition window

3. In the **Application name** area, select **Dosrad** if necessary.
4. Click **Apply** (if necessary).



Dose cartography (or Mapping)

1. Initialize a meshing file (refer to 3.4 “Creating the Meshing”“Initializing the meshing “).

To create a dose cartography:

2. In the meshing tree structure, select the shape (*New General truncated sphere*, for instance) of the geometrical model in which you wish to create a dose cartography.
3. Double-click the shape to edit the meshing parameters.

The following window opens:

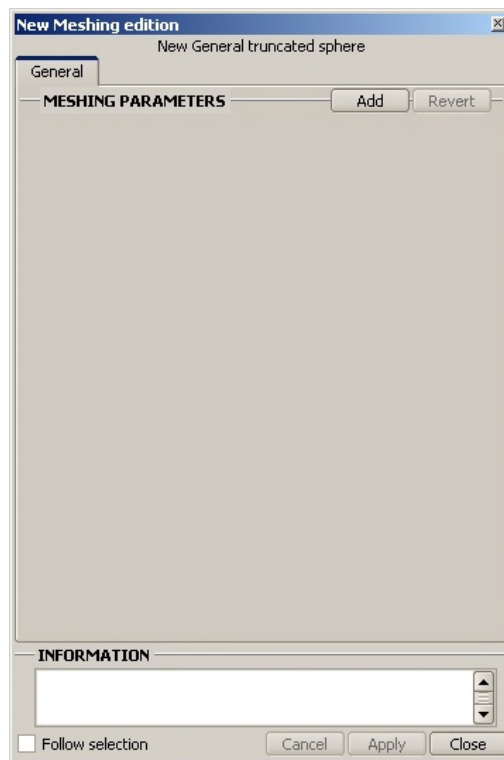


Figure 30 - Mapping meshing edition

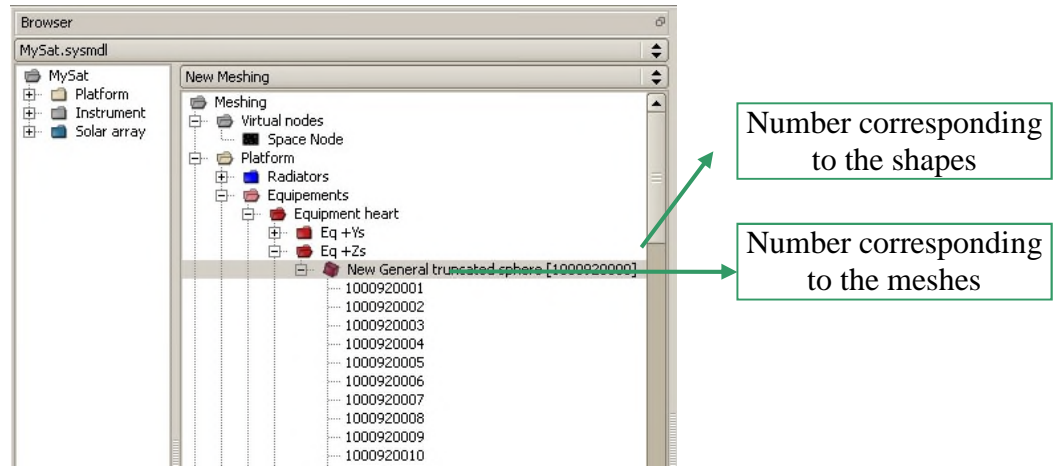
4. In the **Meshing parameters** area, click **Add** and define a meshing for the shape (for instance a 10/20/1 meshing: a, b and c set to 10/20/1).



5. Click **Apply**.



In the browser, the system creates the elements with an arbitrary numbering.

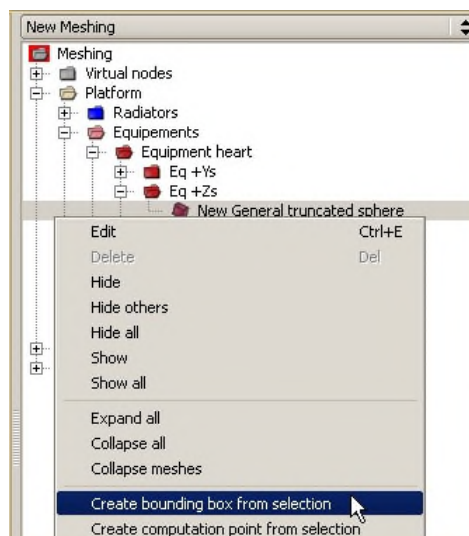


You can now save the meshing file you have just created.

6. In **File -> Meshing -> Save as** menu, save your meshing file as *mapping.sysmsh*.

Bounding box

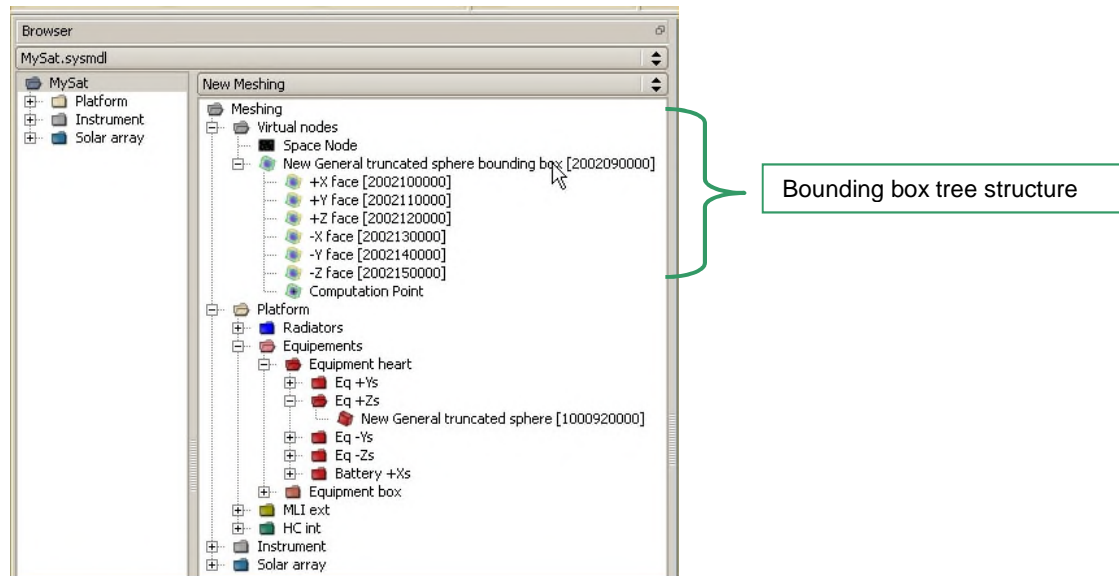
1. Initialize a meshing file (refer to 3.4 “Creating the Meshing”/“Initializing the meshing “).
 To create a bounding box from selection:
2. In the meshing tree structure, select the shape (*New General truncated sphere*, for instance) of the geometrical model in which you wish to create a bounding box.
3. Via the right click menu, select **Create bounding box from selection**.



Result: The bounding box is created with an arbitrary numbering. In the browser window, the bounding box is positioned under **Virtual nodes**. Its name finishes with “bounding



box". One computation point is systematically added to this bounding box, located at the center and with the same referenced shapes.



4. In the meshing tree structure, double-click the bounding box to edit the meshing parameters.

The following window opens:

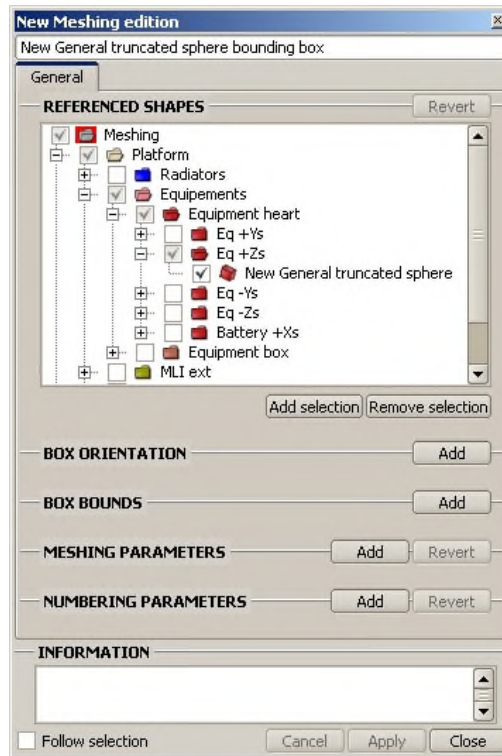




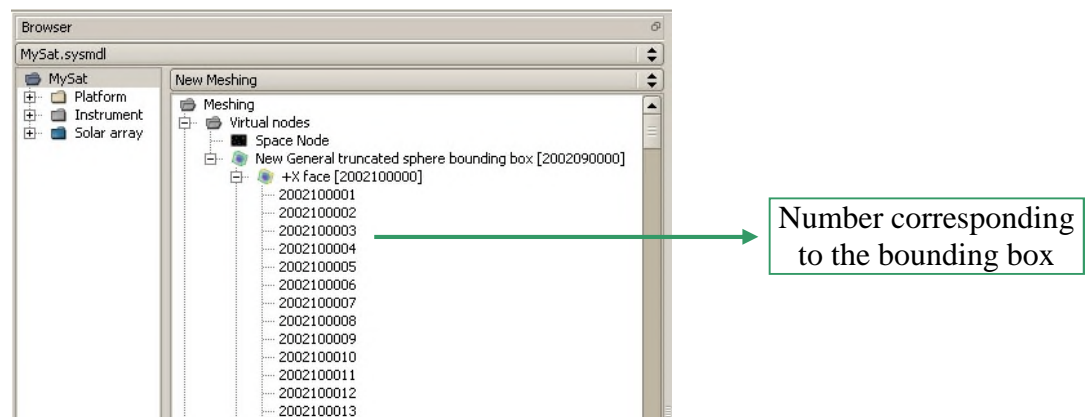
Figure 31 - Bounding box meshing edition

- In the **Meshing parameters** area, click **Add** and define a meshing for the shape (for instance a 20/20/20 meshing: a, b and c set to 20/20/20).



- Click **Apply**.

In the browser, the system creates the elements with an arbitrary numbering.



You can now save the meshing file you have just created.

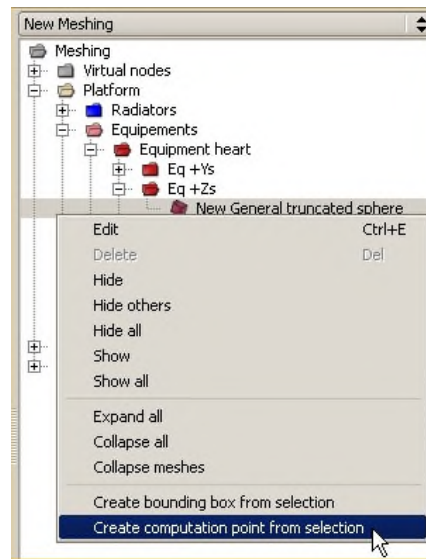
- In **File -> Meshing -> Save as** menu, save your meshing file as *boundingbox.sysmsh*.

Computation point

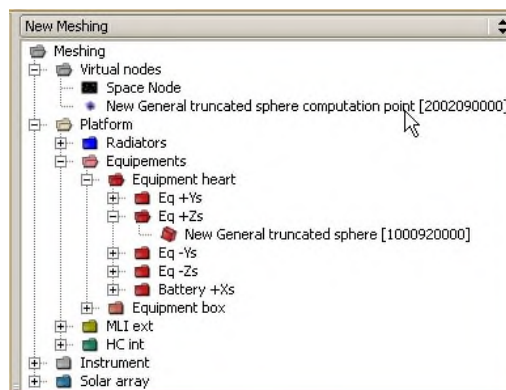
1. Initialize a meshing file (refer to 3.4 “Creating the Meshing”/“Initializing the meshing”).

To create a computation point from selection:

2. Select the shape of the geometrical model that will become the referenced shape attached to the created computation point.
3. Via the right click menu, select **Create computation point from selection**.



Result: The computation point is created with an arbitrary numbering. In the browser window, the computation point is positioned under **Virtual nodes**. Its name finishes with “computation point”.



Computation point

- In the meshing tree structure, double-click the computation point to edit its position.

The following window opens:

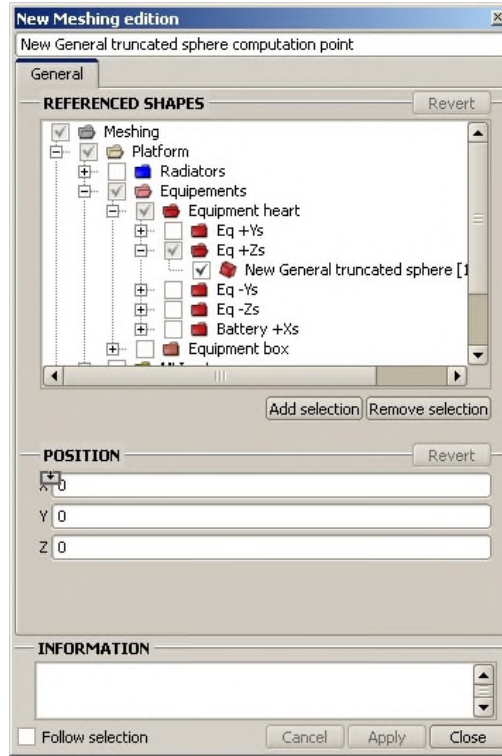
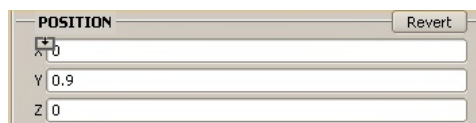


Figure 32 - Computation point meshing edition

- In the **Position** area, define a 0/0.9/0 position (X, Y and Z set to 0/0.9/0).



- Click **Apply**.

In the 3D view, the system displays the new coordinates of the computation point.

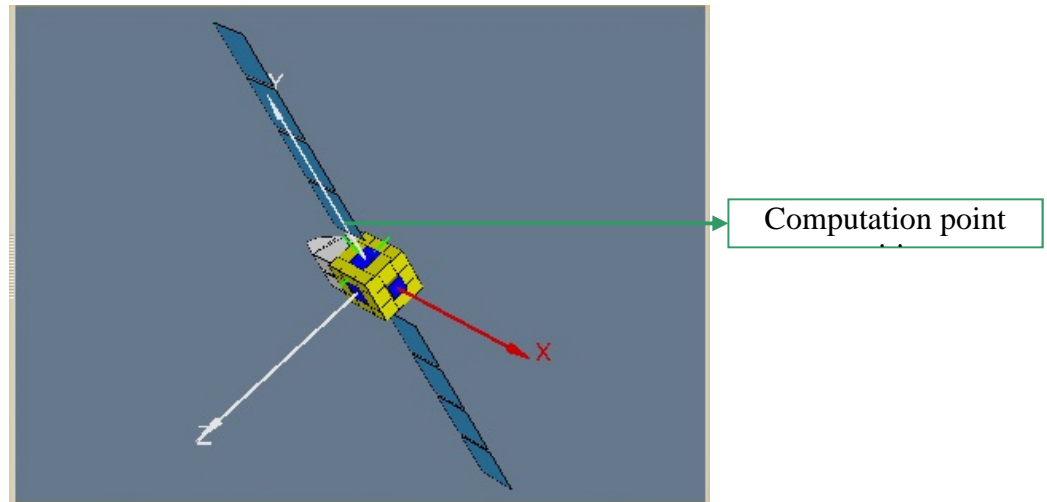


Figure 33 - Computation point position

You can now save the meshing file you have just created.

7. In **File -> Meshing -> Save as** menu, save your meshing file as *computpoint.sysmsh*.



The surface meshing (used for dose cartography), the bounding box and the computation point can be created from the same meshing file.

Numbering setting

SYSTEMA applies a default numbering sufficient for radiation analyses. It is not necessary to set numbering parameters. Furthermore, since version 4.4.2, it is not possible to modify the numbering anymore.



14.5 Creating the Mission

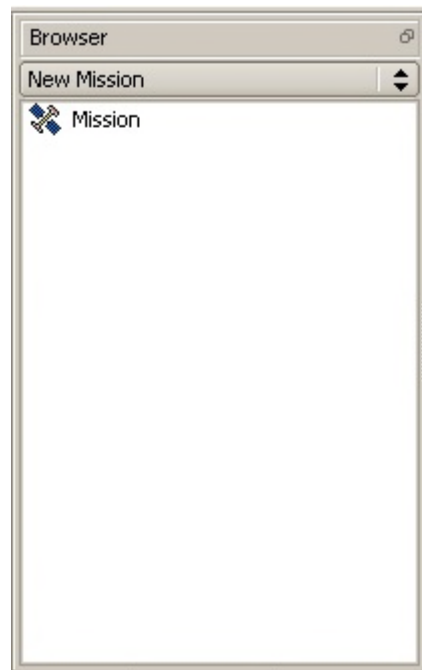
In this section, you will set all the parameters for the mission. Since version 4.4.2, this part can be skipped, as a meshing file can be attached directly to the processing diagram.

Creating a new mission

To create the mission, follow these steps:

1. Open the **Mission** tab in **SYSTEMA**.
2. Click the **File** -> **New** menu.

The Mission browser displays on screen:



3. In the browser, double-click **Mission**.



The edition window opens:

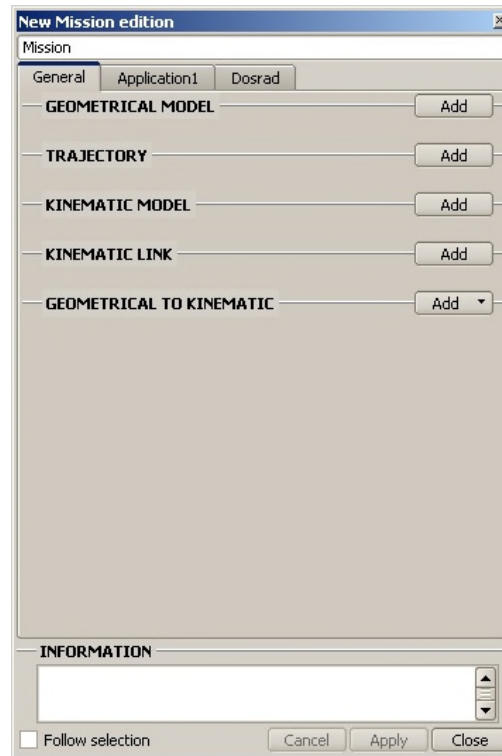


Figure 34 - New Mission edition window

4. Click the **General** tab.
5. In the **Geometrical model** area, click **Add** and then **Other...** to select the *MySat.sysmdl* file.
6. Click **Apply**.



In the 3D view, the system displays the geometrical model in the **Mission** tab

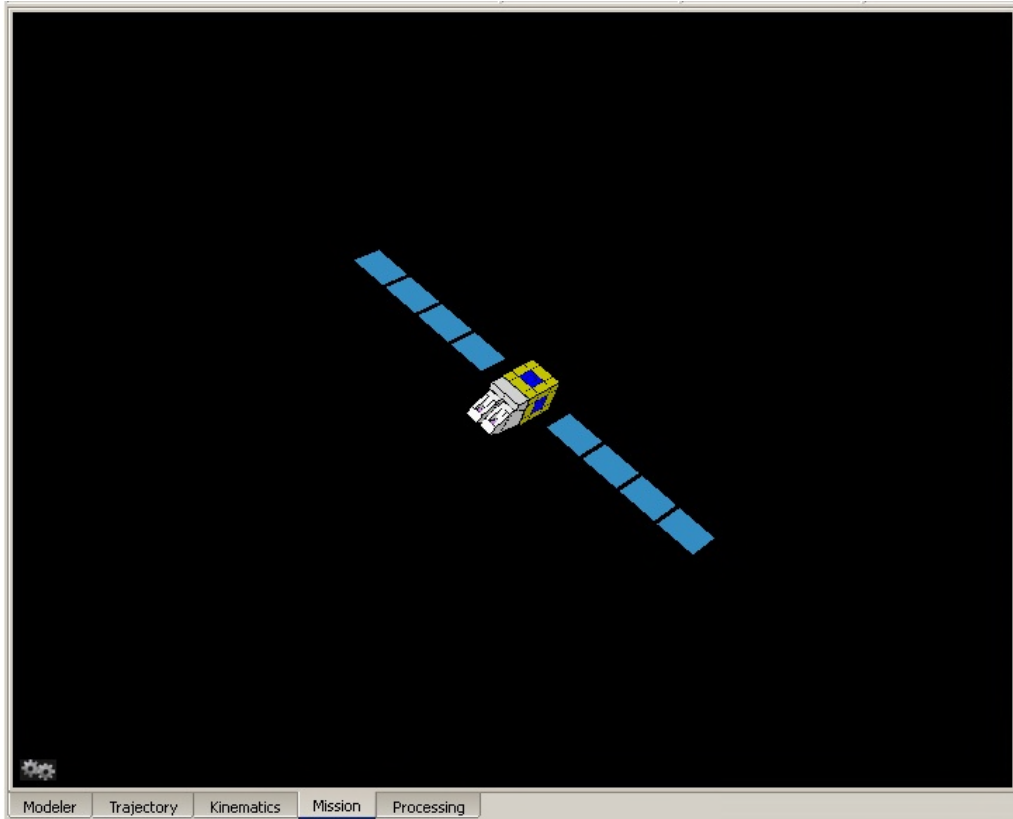


Figure 35 - Mission

Now, you must associate the meshing model with the mission.

Linking the meshing

At this stage, the previously created meshing must be associated with the mission.

To link the created meshing:

1. In the **New Mission edition** window, select the **DOSRAD** tab.



The following window displays:



Figure 36 - Dosrad tab

2. Click **Add** in the **Meshing** area.
3. Click **Other...** to select the meshing file (*mapping.sysmsh*, *boundingbox.sysmsh* or *computpoint.sysmsh*) you have created in the previous section.



Do not enter the COMPUTATION DATES.

4. Save the mission file.



14.6 Defining a Computation Processing Diagram

In this section, you will create a diagram for computing the results based on **Mapping**, **BoundingBox** and **ComputPoint** modules.

Creating a processing diagram

To create a processing diagram:

1. Select the **Processing** tab in the **SYSTEMA** window.
2. Click the **File -> New** menu.
3. Click **DOSRAD**.

The processing diagram is displayed.

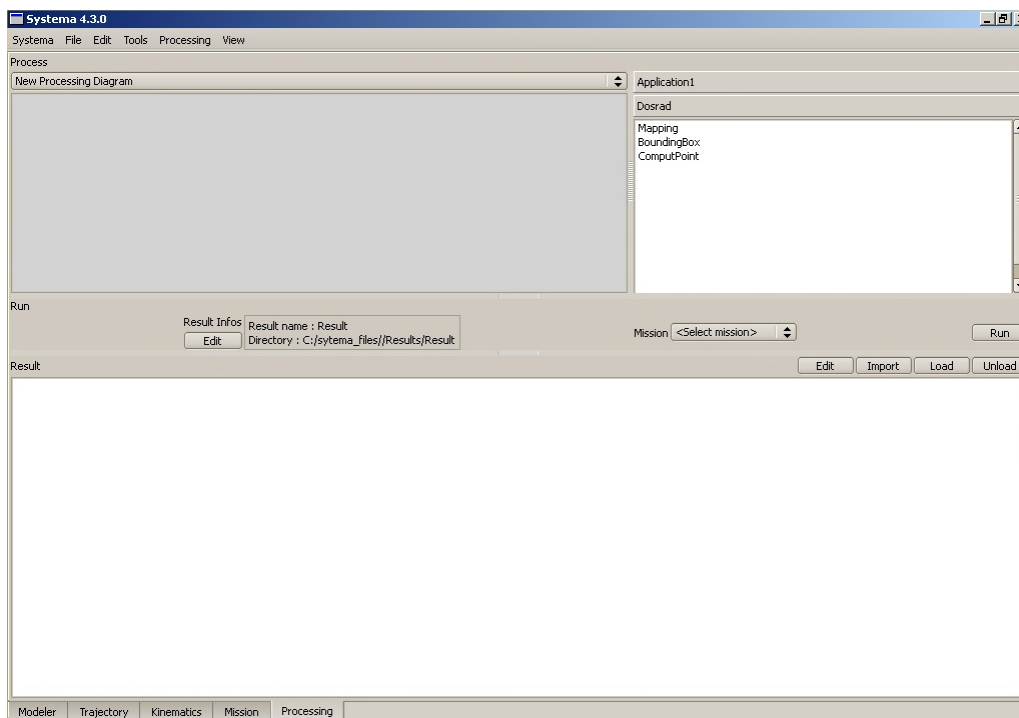


Figure 37 - Processing diagram

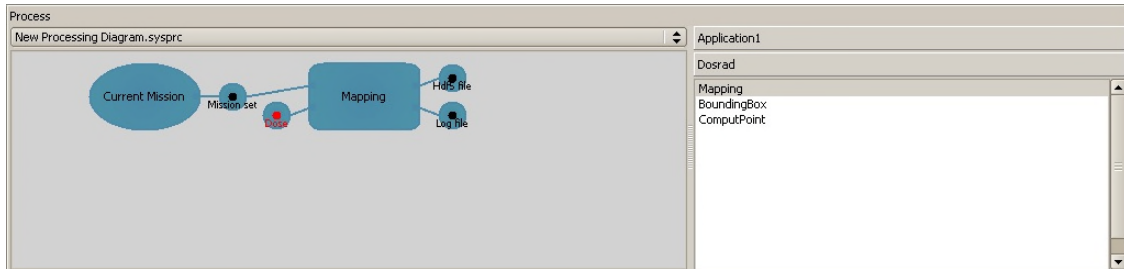
You can set each module in the **DOSRAD** process according to the wished results.

Mapping module



In the **DOSRAD** area, double click **Mapping**.

The following processing diagram is displayed:

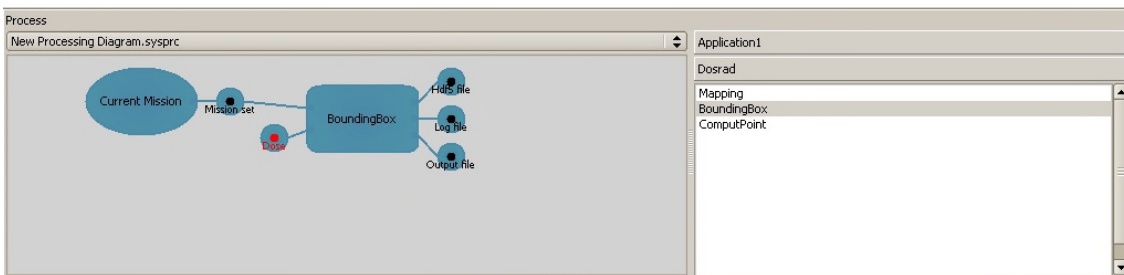


BoundingBox module



In the **DOSRAD** area, double click **BoundingBox**.

The following processing diagram is displayed:

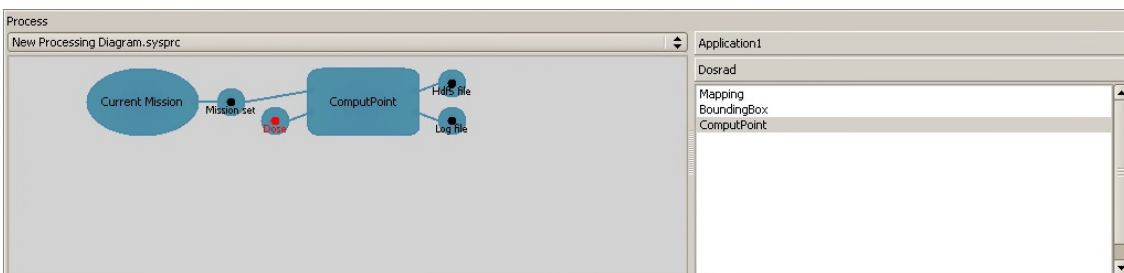


ComputPoint module



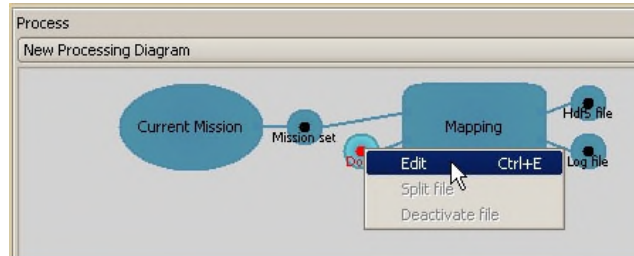
In the **DOSRAD** area, double click **ComputPoint**.

The following processing diagram is displayed:



Activating the dose curve file

6. Via the right click menu of the dose input file, select **Edit**:



The dose curve file edition is displayed:

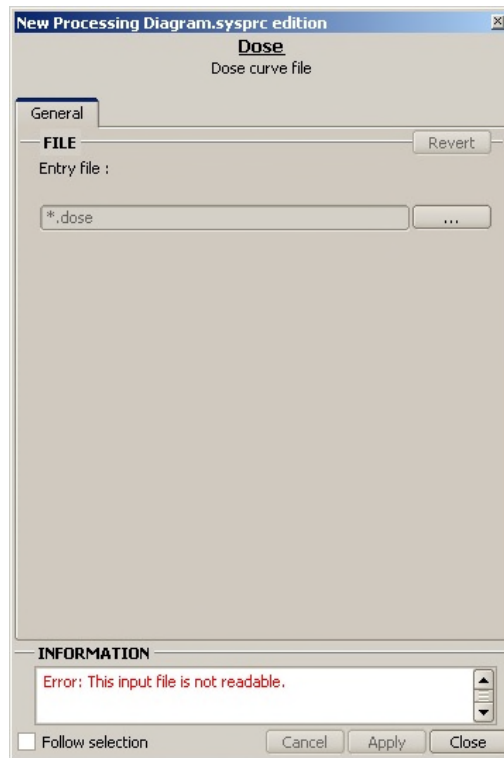



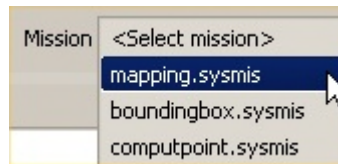
Figure 38 - Dose curve file edition

7. Click  to select the test.dose file.
8. Click **Apply** to activate the dose curve file.



Running the computing files

9. Double click the Current Mission, select the wished mission (mapping.sysmis, boundingbox.sysmis or computpoint.sysmis) or select "Meshing" in the "Type of exported structure" field and select your meshing file:



If you do not select the mission or meshing, a warning displays and the results cannot be computed.

10. Save the process diagram.

11. Click .



The results are saved as per the information in the *Result infos* area. The files created within these results are the result files generated from the diagram and a .sysres file corresponding to the data displayed on the diagram.

If you want to display the results on different processes in the *Result* area, be careful not to overwrite the sysres. file when you generate a result file.

We recommend to change the result file name for each process.

You have now created a complete mission and generated the result files.



To learn more about result management, see the Systema documentation.



14.7 Results

After the run, results and log files are generated. Some of them are created by **SYSTEMA**. These files are:

- *.sysset: mission synthesis file for **DOSRAD** (xml format):

It contains the description of the model and the meshing and it is an input of all the computation modules.

- log.txt (ascii format): information and control file:

It describes the execution of the launched process.

- *.sysres: results synthesis file (xml format).

The results and log files that are generated by each computation module of **DOSRAD** are described below:

- *.box.log (bounding box) or *.pt.log (computation point) or *.map.log (dose cartography): information and control file (ascii format):

It contains the description of the run parameters and model information.

- *.box.h5 or *.pt.h5 or *.map.h5: results file (hdf5 format):

it is used by the post-processing of **SYSTEMA** for the 3D view of results and it can also be loaded into an HDF explorer. The file is structured in three directories:

- **Box meshes or Mapping meshes or Computation points:**

Information on meshes (area, number, name) or computation points (number, name).

- **Rays:**

Information on rays launched from meshes or computation points (effective launched rays, number of rays that encountered surfaces with a thickness equal to zero, number of rays that did not encounter surfaces).

- **Results:**

Dose (in rad) and thickness (in mm) results for each mesh or computation point with the selected technique.

- *.box.out(bounding box) or *.pt.out (computation point): results file (ascii format):

It is created by the bounding box module and contains the results synthesis (aluminium equivalent thickness per bounding box face, total dose and total equivalent thickness per bounding box).

How to display results in a 3D view

We describe below the procedure to display 3D results for a bounding box computation.



1. Open the **Modeler** tab in **SYSTEMA**.
2. In the 3D view, via the right click menu, select **Configuration window**.
3. Select the result file and the property (for example: Dose per face) you wish to display.
4. Close the configuration window.

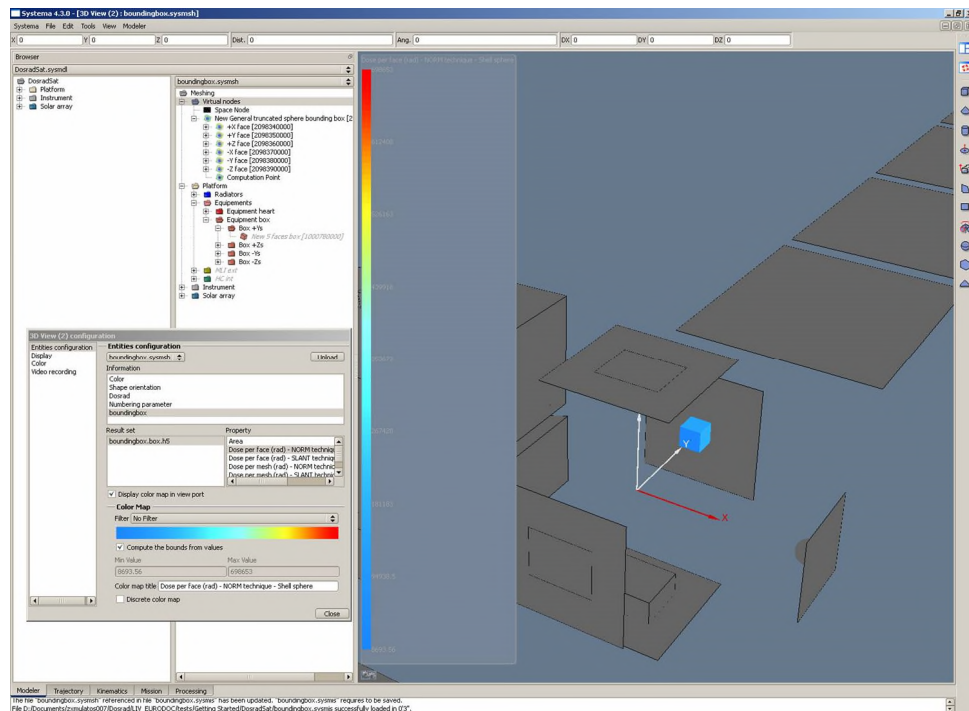


Figure 39 - Dose results per face

Among the available properties, you can select:

- Area (information per bounding box mesh).
- Number of (thick=0) rays per mesh.
- Number of launched rays per mesh.
- Number of void rays per mesh.



- Dose (result for each bounding box):
 - **Per mesh**
 - **Per face**
 - **Sub per face (if SLANT technique is chosen, results for Sub-contractors are also given)**
 - **Total and total sub**
- Thickness (result for each bounding box):
 - **Per mesh**
 - **Per face**
 - **Sub per face**
 - **Total and total sub**
- Mean thickness



The names of the results in the Property list are followed by the technique and the type of geometry selected. For instance, SLANT technique – Solid sphere.

